VisualCAM

2006

User Manual

WISE SOFTWARE

©2006-2007 WISE Software Solutions, Inc. All rights reserved.

WISE Software Solutions, Inc.
2700 East Ninth Street
Suite 100
Newberg, OR 97132

This information is copyrighted; all rights are reserved by WISE Software Solutions, Inc. This information may not, in whole or in part, be copied, photocopied, reproduced, translated or reduced to any electronic medium or machine-readable form without the prior written consent of WISE Software Solutions, Inc.

GerbTool, VisualCAM, WISE Software, and WISE Software Solutions are trademarks of WISE Software Solutions, Inc.

DXF and AutoCAD are registered trademarks of Autodesk, Inc.
CAM350 and FabFactory are trademarks of Downstream Technologies LLC.
DirectCAM, PADS, and PowerPCB are registered trademarks of Mentor Graphics.
ViewMaster is a trademark of PentaLogix LLC.
ODB++ is a trademark of Valor Computerized Systems.
FATF is a trademark of UGS Corp.
FLEXlm is a registered trademark of Macrovision Corporation.
Microsoft, Windows, Windows NT, Windows 2000, and Windows XP are registered trademarks or trademarks of the Microsoft Corporation in the United States and other countries.

All other product and company names herein are the trademarks of their respective owners.

Table of Contents

About VisualCAM **1**
System Requirements 1

Contacting WISE Software **2**
Headquarters 2
Technical Support 2

Main VisualCAM Window **3**
Title Bar 3
Menu Bar 3
Toolbar Button Reference 3
Layer Bar 6
The Navigator 7
Data Tab 7
Commands Tab 27
Selection Filter 29
Item Properties Display 30
Status Bar 34
Workspace 35
XY Bar 35
Aerial Bar 35
Color Bar 36

Hotkeys **38**

Mouse and Function Key Commands **40**
Mouse Functions 40
Function Keys 40

Dialog Boxes **41**
File Selection 41
Edit Forms 41
Accessing Help 42

Performance Tips **43**
Hotkeys Execute Immediately 43
Interrupting Redraws and Highlights 43
Undoing Edits 43
Using the Selection Filter 43
Copying to the Windows Clipboard 43
Programming Your Mouse and Function Keys 43
Memory Considerations 43

Tutorials **44**
Using Files 44
Gerber for Beginners 47
Working with Apertures 57
Working with Composites 60
Performing DRC & DFF Analyses 64
Comparing Netlists 65
Panelization 67
Creating Test Fixture Files 69
Working with Embedded Passives 72
Creating Stencils 73
Reverse Engineering for Assembly 74

VisualCAM Command Reference **77**
File Menu 77
File|New 77
File|Open 77
File|Close 78
File|Merge 78
File|Save 79
File|Save As 79
File|Import 79

File|Export 94
File|Page Setup 101
File|Print 102
File|Print Preview 103
File|Print Setup 103
File|Send 103
File|Save Log 103
MRU File List 103
File|Exit 103
Edit menu 104
Edit|Undo 104
Edit|Redo 104
Edit|Select 105
Edit|Copy Selection To Clipboard 106
Edit|Paste Clipboard 106
Edit|Item 106
Edit|Copy 108
Edit|Move 109
Edit|Delete 110
Edit|Text 111
Edit|Vertex 111
Edit|Clip 112
Edit|Join 113
Edit|Rotate 113
Edit|Mirror 114
Edit|Scale 114
Edit|D-Code 115
Edit|Align Layers 116
Edit|Snap Pads 116
Edit|Origin 117
Edit|NC 117
Edit|Purge 122
View menu 123
View|Window 123
View|Zoom In 123
View|Zoom Out 124
View|Pan 124
View|All (Fit) 124
View|Film Box 124
View|Redraw 125
View|Sketch 125
View|Overlay 125
View|Grid 125
View|Composites 125
View|Backside 125
View|Virtual Panel 126
View|Parts 126
View|Clear Highlights 126
View|Highlights 126
View|Selections 126
View|Selection Filter 126
View|Save 126
View|Recall 126
View|Previous 126
View|Navigator 127
View|Toolbars 127
View|Restore Toolbars 127
View|View Tabs 127
Add Menu 129
Add|Flash 129
Add|Draw 129
Add|Rectangle 130

Add|Vertex 130
Add|Circle 130
Add|Arc Ctr 131
Add|Arc 3 Pt 131
Add|Array 131
Add|Polygon 132
Add|Text 133
Add|Drill 134
Add|Slot 134
Add|Mill Path 134
Add|NC Circle 135
Add|Drilled Text 136
Add|Break Tab 137
Add|Operator Message 138
Add|Optional Stop 138
Setup Menu 139
Setup|Layers 139
Setup|Apertures 140
Setup|Composites 144
Setup|Layer Sets 145
Setup|NC Tools 146
Setup|Break Tabs 148
Setup|Stencil Shapes 150
Setup|Footprint Library 150
Documentation Menu 151
Documentation|Reports 151
Documentation|Redline 153
Documentation|Drawing 157
Analysis Menu 164
Analysis|DRC/MRC 164
Analysis|Netlist Compare 168
Analysis|Layer Compare 168
Analysis|Find Duplicates 169
Analysis|Copper Area 169
Query Menu 170
Query|Item 170
Query|Net 170
Query|User Data 171
Query|Embedded Passive 171
Query|Highlight 172
Query|Measure 172
Query|Extents 173
Options Menu 174
Options|Grid Snap 174
Options|Ortho Line Snap 174
Options|Arcs 360 174
Options|Units/Precision 174
Options|Configure 174
Options|Customize Toolbar 178
Options|Import Settings 179
Macro Menu 181
Macro|Run 181
Macro|Load 181
Macro|Developer 181
Macro|Record 181
Tools Menu 182
Tools|Panelize 182
Tools|Netlist 187
Tools|Solder Mask 188
Tools|Paste Mask 190
Tools|Snoman 193
Tools|Teardrops 194

Tools|Fix SilkScreen ... 194
Tools|Pad Removal ... 195
Tools|NC ... 195
Tools|Test Points ... 200
Tools|Embedded Passives ... 204
Tools|Layer Spread ... 207
Tools|Stencils ... 207
Tools|Convert ... 209
Assembly Menu ... 218
Assembly|Setup Footprint Library ... 218
Assembly|Parts ... 218
Assembly|Fiducials ... 224
Assembly|Import BOM ... 224
Assembly|Export FATF ... 225
User menu ... 226
Custom Aperture Editor ... 227
Main Menu ... 227
File Tool Bar ... 227
View Tool Bar ... 227
Coordinate Display ... 227
Graphics Tool Bar ... 227
Status Display ... 227
Custom Aperture Editor Command Reference ... 228
File|Save ... 228
File|Exit ... 228
Edit|Undo ... 228
Edit|Copy ... 228
Edit|Move ... 228
Edit|Delete ... 229
Edit|Change Drill Tools ... 229
View|Window ... 229
View|Zoom In ... 229
View|Zoom Out ... 230
View|Pan ... 230
View|All ... 230
View|Redraw ... 230
View|Sketch ... 230
View|Toolbar ... 230
View|Status Bar ... 231
View|View Bar ... 231
Add|Circle ... 231
Add|Line ... 231
Query|Item ... 231
Options|Grid ... 231
Options|Metric ... 232
File Format Technical Reference ... 233
Aperture List Files ... 233
ACR Files ... 235
Text Font Files ... 241
Sample VisualCAM Netlist (.nl) File ... 242
Color List File ... 243
Glossary ... 244
Index ... 249

About VisualCAM

VisualCAM provides electronic design and manufacturing engineering professionals with the tools they need for complete control over their PCB design, manufacturing, and assembly data flow. From visual verification to high-level data analysis, VisualCAM simplifies and automates your PCB post-processing tasks. It includes everything from feature-rich data editing to automated Assembly Reverse Engineering -- all of which are there to streamline the flow of data between PCB design, manufacturing, and assembly.

- VisualCAM's intuitive graphical user interface, Navigator, toolbars, and hotkeys allow you to focus on accomplishing tasks quickly and efficiently, rather than on the technical details of operating the software.
- Native support of ODB++, ODB++ XML, and IPC-2581 (Offspring) drive the flow of intelligent data from any major CAD system to VisualCAM.
- When intelligent data is unavailable, the Import Wizard takes the guesswork out of importing traditional Gerber files into VisualCAM.
- Validating the connectivity of any design is fast, accurate, and efficient using the HyperNETLIST Generation and Comparison functions. During netlist comparison, a Pin-Point Error feature eliminates the headaches involved with troubleshooting shorts and opens.
- Dynamic DfM Analysis provides a full suite of analysis tools that are presented in a single, checklist-style environment. This "single run" setup processes all your rule combinations at one time, thus increasing throughput and eliminating repetitive processing.
- The efficiency of VisualCAM's automated tools, such as the Solder and Paste Mask Generators/Optimizers, Advanced Panelization, and Bare-Board Test Fixture Generation, reduces your time-to-market and costs.
- Automated Assembly Reverse Engineering capabilities offer state-of-the-art extraction of component footprint and centroid information from non-intelligent Gerber data. No other software tool offers the same level of automation and performance found in VisualCAM.

System Requirements

The following minimum system configuration is required for running and installing VisualCAM.

- IBM PC compatible with at least a 1GHz Pentium CPU.
- Windows 2000, XP or later.
- For best graphics display, a 17" or larger monitor with at least 1280 x 1024 resolution is recommended.
- At least 512 megabytes (MB) of RAM and 30 MB of hard disk space.

Contacting WISE Software

If you have any questions regarding GerbTool or WISE Software Solutions, feel free to contact us.

Headquarters

Address:

WISE Software Solutions, Inc.
2700 E. 9th Street, Suite 100
Newberg, OR 97132 USA

Phone:

(503) 554-8855

FAX:

(503) 554-1220

E-Mail:

North American Sales sales@wssi.com

International Sales int_sales@wssi.com

World Wide Web:

http://www.wssi.com

Technical Support

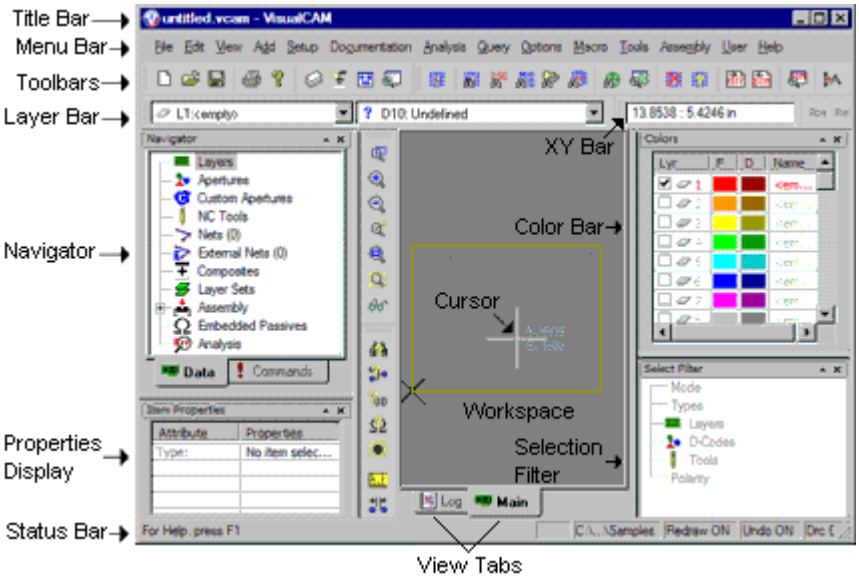
If you purchased VisualCAM from an authorized reseller, you should contact them directly for technical support as they will have a better understanding of your needs and intended uses of the software.

If you are unable to get satisfactory assistance in solving your problems, you may contact WISE Software Solutions using the following methods:

- Check our web site at http://www.wssi.com/visualcam for up-to-date FAQ's with video demos, as well as access to the VisualCAM User Forum.
- Send technical questions by e-mail to tech@wssi.com. Send license requests to licenses@wssi.com.
- Send a FAX message to (503) 554-1220.
- Call (503) 554-8855 between the hours of 8:00 A.M. and 4:00 P.M., Pacific Time.

Main VisualCAM Window

The main VisualCAM window, or "desktop", is illustrated below. All toolbars and control bars may be moved to a location you prefer by clicking on the bar and dragging it to a new location. They may be docked to an edge of the VisualCAM window, or float in a small window. Most of the items in the window can also be resized or closed so that you can customize the space to suit your needs.



Title Bar

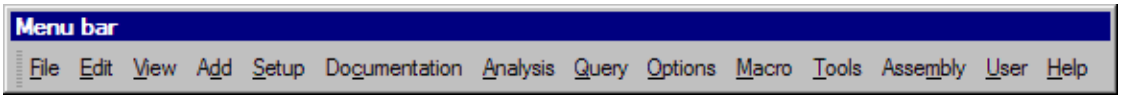
The Title Bar provides the filename of the currently loaded database, and the VisualCAM product name. There are three buttons on the right side of the Title Bar:

Clicking the Minimize button reduces the desktop to a button on the Windows task bar.

Clicking the Maximize button makes the desktop fill the entire screen. The button's appearance then changes to an image of two windows (the Restore button). Clicking this restores the window to its previous size.

Clicking the Close button exits the VisualCAM program. This functions the same as the File|Exit command.

Menu Bar



To view a menu, position your cursor on the Menu bar, and click on it with the left mouse button. This presents lists of commands that you can execute by clicking on them individually.

Terminating a Command

You may terminate a command, or at least one level of a multi-step command, by pressing the Esc key on your keyboard. You may also right-click and choose Cancel from the right-click shortcut menu. If you would like to terminate a command using the right mouse button, you can disable the right click shortcut menu. See the Options|Configure command.

Selecting another command from the Menu bar will also terminate any active command. Hotkeys, however, will not terminate an active command.

Toolbar Button Reference

Each button within the toolbar represents a shortcut to a menu command. When you click on a toolbar button, the command associated with that button is invoked. You can control which toolbars appear in the window with the View|Toolbars command. You can change which button appears in each toolbar by using the Options|Customize Toolbar command.

Drawing Buttons

These buttons are used for working with Drawing layers.

Documentation|Drawing|Dimensioning|Add Dimension

Documentation|Drawing|Dimensioning|Delete Dimension

Documentation|Drawing|Dimensioning|Modify Dimension

 Documentation|Drawing|Dimensioning|Add Line
 Documentation|Drawing|Dimensioning|Delete Line
 Documentation|Drawing|Dimensioning|Modify Line
 Documentation|Drawing|Drill|Add Hole Chart
 Documentation|Drawing|Drill|Delete Hole Chart
 Documentation|Drawing|Drill|Modify Hole Chart
 Documentation|Drawing|Drill|Update Hole Chart
 Documentation|Drawing|Fabrication|Add Note Balloon
 Documentation|Drawing|Fabrication|Delete Note Balloon
 Documentation|Drawing|Fabrication|Modify Note Balloon

Drill Buttons

These buttons are used for working with NC data.

 Add|Drill
 Add|Slot
 Add|NC Circle|Drilled
 Add|Drilled Text
 Add|Mill Path
 Edit|NC|Path Properties
 Edit|NC|Reverse Path Direction
 Edit|NC|Explode
 Add|NC Circle|Milled|CCW Inside
 Add|NC Circle|Milled|CCW Outside
 Add|NC Circle|Milled|CW Inside
 Add|NC Cicle|Milled|CW Outside
 Add|Operator Message
 Edit|NC|Delete|Operator Message
 Edit|NC|Change|Operator Message
 Add|Optional Stop
 Edit|NC|Delete|Optional Stop
 Setup|Break Tabs
 Add|Break Tab
 Edit|NC|Delete|Break Tab
 Edit|NC|Move Break Tab
 Edit|NC|Change|Break Tab
 Edit|Change NC Tools
 Tools|Convert|Gerber to NC
 Tools|NC|Set Order
 Tools|NC|Optimize
 Tools|NC|Display Settings
 Setup|NC Tools
 Documentation|Reports|NC Tools
 File|Export|NC (Drill/Mill)

Edit Buttons

These buttons are used for working with graphics.

 Edit|Undo
 Edit|Redo
 Edit|Copy
 Edit|Move
 Edit|Delete
 Add|Flash
 Add|Draw
 Add|Rectangle
 Add|Circle
 Add|Arc Ctr
 Add|Arc 3 Pt
 Add|Array
 Edit|Vertex|Add & Add|Vertex
 Edit|Vertex|Delete
 Add|Polygon
 Edit|Rotate
 Edit|Mirror
 Edit|Align Layers
 Edit|Clip
 Edit|Join
 Edit|Origin
 Edit|Scale
 Edit|D-Code|Transcode
 Edit|D-Code|Polarity
 Edit|D-Code|Scale
 Edit|D-Code|Explode Customs
 Edit|Item
 Edit|Select|New Group

 Edit|Vertex|Move
 Edit|Vertex|Segment Delete
 Add|Text
 Edit|Text
 Edit|Select|Add To
 Edit|Select|Remove From
 Edit|Select|Clear
 Edit|Select|Invert

Mainframe Buttons

These buttons are used for working with files, and defining layers and apertures.

 File|New
 File|Open
 File|Save
 File|Print
 Help|About VisualCAM
 Setup|Layers
 Setup|Composites
 Setup|Apertures
 Documentation|Reports|Apertures

Parts Buttons

The Parts buttons are for assembly reverse engineering of parts data.

 Assembly|Setup Footprint Library & Setup|Footprint Library
 Assembly|Parts|Identify Using|Footprint Library
 Assembly|Parts|Identify Using|IPC-D-356 Data
 Assembly|Parts|Identify Using|SilkScreen Layer
 Assembly|Parts|Identify Using|Centroid File
 Assembly|Parts|Identify Using|Select Footprint
 Assembly|Fiducials|Identify
 Assembly|Fiducials|Report
 Assembly|Parts|Delete
 Assembly|Parts|Query
 Assembly|Export FATF
 Assembly|Import BOM
 Assembly|Parts|BOM Report & Documentation|Reports|BOM
 Tools|Convert|Drawn Text

Query Buttons

These buttons give you quantitative information about specific database items.

 Query|Item
 Query|Net
 Query|User Data
 Query|Embedded Passive &
Tools|Embedded Passives|Query
 Query|Highlight
 Query|Measure|Point To Point
 Query|Measure|Edge To Edge
 Query|Measure|Center To Center
 Analysis|Copper Area
 Query|Extents

Redline Buttons

These buttons are used for adding comments and other information which are stored separately from the layer design information.

 Documentation|Redline|View Redlining
 Documentation|Redline|Add Text
 Documentation|Redline|Add Balloon Text
 Documentation|Redline|Add Arrow
 Documentation|Redline|Add Line
 Documentation|Redline|Sketch

Documentation|Redline|Delete
Documentation|Redline|Properties

Settings Buttons

These buttons allow you to control display properties and other program settings.

View|Sketch
View|Overlay
View|Composites
View|Selection Filter
View|Grid
Options|Grid Snap
Options|Arcs 360
Options|Ortho Line Snap
Options|Units/Precision

Tool Buttons

These buttons execute CAM and Analysis utilities.

Tools|Panelize|Advanced
Tools|Panalize|Simple
Analysis|DRC/MRC
Tools|Snoman
Tools|Teardrops
Tools|Fix Silkscreen
Tools|Pad Removal|Isolated
Tools|Convert|Arcs To Segmented Arcs
Tools|Convert|Segmented Arcs To Arcs
Tools|Layer Spread
Tools|Netlist|Generate
Tools|Convert|Drawn Pads|Select Draws
Tools|Convert|To Custom
Tools|Convert|Drawn Pads|Automatic

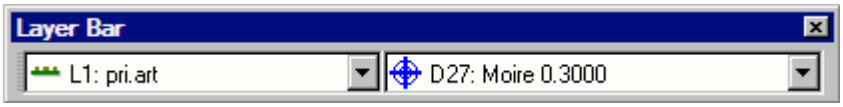
View Buttons

These buttons manipulate your view of the data in the workspace.

View|Window
View|Zoom In
View|Zoom Out
View|Pan
View|All (Fit)
View|Film Box
View|Redraw

Layer Bar

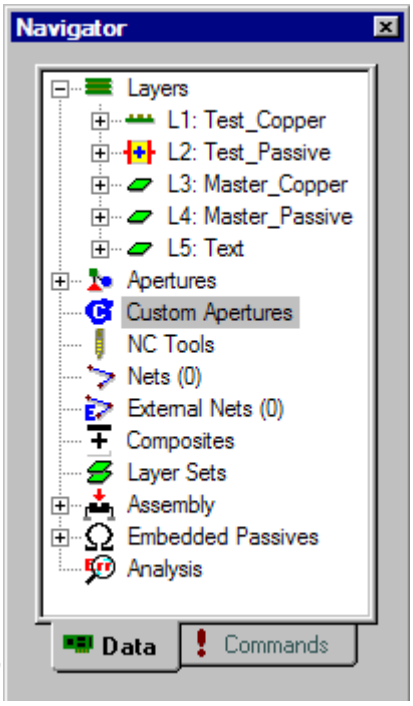
The Layer Bar displays the active layer on the left. The graphic next to the layer number indicates there is data on the layer, and what type it is. You can make a different layer active by selecting it from this list.



On the right, if the active layer is *not* an NC layer, the active D-code is displayed. The graphic next to the D-code number shows the shape of each aperture. All custom apertures are signified by an irregular shape with a "C" in the middle (the actual shape is not indicated.) You can make a different D-code active by selecting it from this list.

If the active layer is an NC layer, the active NC tool is displayed. If the Add|Break Tab command is in use, the currently active tab is displayed, with an icon that indicates its type. You can make a different tool or tab active by selecting it from this list.

The Navigator



The Navigator provides a quick-reference list of your database elements, and another way of accessing VisualCAM command functions. Most of the Navigator functions, such as changing layer names, adding apertures, and printing analysis reports, are controlled with a right-click shortcut menu.

You can dock the Navigator anywhere in the main VisualCAM window by clicking on the top of the Navigator bar and dragging it to another location. You can toggle the view of the Navigator on or off by selecting the View|Navigator command. You can also close the Navigator by clicking on the X in the upper-right corner. You can expand the size of the bar to the entire height of the workspace by clicking the up-arrow; then reduce its size by clicking the down-arrow. While it is docked in the VisualCAM window, you can resize the height and width of the bar by moving your cursor to one of its outer edges. When the cursor changes to ⇹, click and hold the left mouse button while you drag the edge to the desired size.

Data Tab

The Data tab provides information about your layers, apertures, NC tools, nets, composites, layer sets and Analysis runs. Functions used for the reverse engineering of parts for assembly, and embedded passive data, are also available. The information is displayed in a tree format, displaying database elements in an expandable/collapsible hierarchy. To expand an area of the tree, click on the plus box (+ icon) next to the desired database element. The "branches" of the information hierarchy are shown, and the plus box becomes a minus box (- icon). To compress or hide the information, click on the minus box. The information hierarchy for that database element is hidden. If there is no Plus symbol next to a topic in the tree, that type of element does not exist in your design, or has been defined but is not used.

You can edit and delete database elements by right-clicking on the various headings and the branches that are associated with each. This not only provides you with shortcuts to functions available in the main menu, but also some functions unique to the Navigator.

Layers

To see a list of layers, click on the plus box next to the Layers heading. The Layers area expands to show all the layers in the design, with their corresponding number and name. To view information about Layer 1, click on the Plus symbol next to L1. The layer name and type is shown. The layer type is also indicated by an icon next to the layer number. For a list of each layer type and their associated icon, see the Color Bar topic.

All the layer information may be modified using the Navigator. To change the draw or flash color used on a layer, use the Color Bar or the Setup|Layers command.

To add a layer

1. Right-click on the Layers heading. A shortcut menu appears.
2. Select the Add Layer command, and the Add New Layer dialog box appears
3. Specify the new layer's Name and Type. If it is an NC layer, select the Tool Table that should be associated with it.
4. Click OK. The new layer appears in the list, though its visibility is turned off.

To delete a layer

1. Right-click on the desired layer in the list. A shortcut menu appears.
2. Select the Delete command. You are asked to confirm the deletion.
3. Click the Yes button to delete the layer.

   -or-

1. Click on the desired layer in the list.
2. Press the Delete key.
3. You are asked to confirm the deletion. Click the Yes button to delete the layer.

To make a layer active

- Double-click on the desired layer in the list. The layer is made visible (if it wasn't already), and becomes the active layer.

  -or-

1. Right-click on the desired layer in the list. A shortcut menu appears.

2. Select the Active command (when a layer is active, a checkmark appears next to the Active command). The layer is now active.

To make a layer visible

1. Right-click on the desired layer in the list. A shortcut menu appears.
2. Select the Visible command (when a layer is visible, a checkmark appears next to the Visible command). The layer is now visible.

   To make a layer invisible, select the command to remove the checkmark.

To view unused layers

1. Right-click on the Layers heading. A shortcut menu appears.
2. Select the Show Unused command, and all unused layers immediately appear in the list.

   To hide unused layers, right-click on the Layers heading. A checkmark appears next to the Show Unused command. Select the command to remove the checkmark and hide the unused layers.

To change a layer's name

1. Click on the plus box next to the desired layer.
2. In the layer detail list, the first item is the layer name. Right-click on the name, and a shortcut menu appears.
3. Select the Rename command. A box appears around the name, indicating that it can be edited.
4. Type in the new layer name, and press the Enter key when you are finished.

To change a layer's type

1. Click on the plus box next to the desired layer.
2. In the layer detail list, the second item is the layer type. Right-click on the type, and a shortcut menu appears. The current layer type has a checkmark next to it.
3. Select the new layer type from the menu.

   If you change the type to an NC layer, a tool table is automatically assigned to it. Its name appears in the layer information in the Navigator. If you want to change the assignment, right-click on the tool table name, select the Change NC Tool Tables command, and the name of the desired tool table.

To move a layer

1. In the Layers list, click on the layer you wish to move, and continue to hold down the mouse button.
2. Move (drag) the mouse cursor until it is over the layer (in the Layers list) that you wish to move the layer to.
3. Release the mouse button. All other layers are reordered, as necessary, so that no empty layers are created and no duplicate layer numbers exist.

To copy all the data from one layer to another

1. In the Layers list, click on the layer you wish to copy from, and continue to hold down the mouse button.
2. Press and hold down the Shift key on your keyboard.
3. Move (drag) the mouse cursor to the layer (in the Layers list) that you want to copy the data to.
4. Release the mouse button and Shift key. The data are immediately copied to the selected layer.

To copy all the data from one layer to multiple layers

1. Right-click on the layer you wish to copy from. A shortcut menu appears.
2. Select the Copy To command. The Select Layers dialog box appears.
3. Select one or more layers to copy the data to, and click OK. The data are immediately copied.

To renumber layers

If you have deleted layers or otherwise have created gaps between the layer numbers in your design, you can renumber the layers from lowest-to-highest and eliminate those gaps.

1. Right-click on the Layers heading. A shortcut menu appears.
2. Click the Compact command. The layers are immediately renumbered.

Apertures

To see a list of apertures, click on the plus box next to the Apertures heading. The Apertures area expands to show all the defined apertures in the design, with their corresponding D-code number and size. The blue graphic next to each aperture shows you the general shape of the aperture. If the aperture is a custom, the name assigned to the custom appears in parentheses. Custom apertures are also listed under the Custom Apertures heading.

All the aperture information may be modified using the Navigator.

To add an aperture

1. Right-click on the Apertures heading. A shortcut menu appears.
2. Select the Add New command, and the New D-code dialog box appears.
3. By default, the lowest available D-code number appears in the text box. Select an unused D-code number, and click OK.

   By default, the aperture is made a round thru-hole and has no size. The aperture will not appear in the Navigator until you use it, or select the Show Unused command from the right-click shortcut menu.

To make an aperture active

1. Right-click on the desired aperture in the list. A shortcut menu appears.
2. Select the Active command (when an aperture is active, a checkmark appears next to the Active command). The aperture is now active.

To highlight all occurrences of an aperture

1. Right-click on the desired aperture in the list. A shortcut menu appears.
2. Select the Highlight command. All occurrences of that aperture, on all visible layers, are highlighted in the workspace.

To view unused apertures

1. Right-click on the Apertures heading. A shortcut menu appears.
2. Select the Show Unused command, and all unused apertures immediately appear in the list.

   To hide unused apertures, right-click on the Apertures heading. A checkmark appears next to the Show Unused command. Select the command to remove the checkmark and hide the unused apertures.

To delete an unused aperture

1. Make sure that unused apertures are displayed (see above).
2. Right-click on the desired aperture in the list. A shortcut menu appears.
3. Select the Spreadsheet command, and the Aperture Setup dialog box appears. The dialog box scrolls to the aperture you selected.
4. Right-click on the aperture, and select the Delete D-code command from the shortcut menu.
5. Click OK to close the dialog box.

To change an aperture's shape

1. Click on the plus box next to the desired aperture.
2. In the aperture detail list, the first item is the D-code shape. Right-click on the shape, and a shortcut menu appears.
3. The current shape has a checkmark next to it. Select the new shape from the menu.

To change an aperture's size

1. Click on the plus box next to the desired aperture.
2. In the aperture detail list, the second item is the D-code size. Right-click on the size, and a shortcut menu appears.
3. Select the Edit command. A box appears around the size, indicating that it can be edited.
4. Type in the new size, and press the Enter key when you are finished.

To change an aperture's type

1. Click on the plus box next to the desired aperture.
2. In the aperture detail list, the third item is the aperture type. Right-click on the type, and a shortcut menu appears.
3. A checkmark appears next to the current type. Select the new type from the list.

To add a flash in the workspace

1. Click on the desired aperture in the Navigator, and hold down the mouse button.
2. Drag the cursor to the desired location in the workspace, and release the mouse button. A new flash is added to the active layer, using the aperture you selected.

Custom Apertures

To see a list of custom apertures, click on the plus box next to the Custom Apertures heading. The Custom Apertures area expands to show all the custom apertures in the design by name. To view a custom aperture's size and the D-codes it's assigned to, click on the plus box next to the desired aperture.

To add a custom aperture

1. Right-click on the Custom Apertures heading. A shortcut menu appears.
2. Select the Add New command, and an undefined aperture with a system default name is added to the Custom Apertures list. You can now edit and rename the new custom aperture.

   -or-

2. To add a custom aperture from an external library file, select the Load From Lib command from the shortcut menu.
3. Select the desired .vlb or .vcam library file, and click OK.
4. You are presented with a list of custom apertures in that library file. Select the desired custom aperture, and click OK. If a custom aperture of the same name already exists, you are warned that the incoming custom aperture will be renamed.

To highlight all occurrences of a custom aperture

1. Right-click on the desired custom aperture in the list. A shortcut menu appears.
2. Select the Highlight command. All occurrences of that custom aperture, on all visible layers, are highlighted in the workspace.

   -or-

1. If a custom is assigned to more than one D-code, click on the plus box next to the desired aperture, to view the D-codes it's assigned to.
2. Right-click on the desired D-code number, and select Highlight from the shortcut menu. All occurrences of that custom aperture D-code, on all visible layers, are highlighted in the workspace.

To edit a custom aperture

1. Right-click on the desired custom aperture in the Custom Apertures area. A shortcut menu appears.
2. Select the Edit command, and the Custom Aperture Editor appears.

   -or-

2. If you want to just increase or decrease the size of a custom aperture by applying a scale factor, select the Scale command from the shortcut menu.
3. You are prompted for a scale factor. The X-Size and Y-Size values are multiplied by this number (you cannot enter zero or negative numbers). For example, if you have a custom with an X and Y size of .06, a scale factor of 2.0 increases the sizes to .12. If you choose a scale factor of .5, the sizes decreased to .03.

To flatten a custom aperture

Custom apertures are often made up of a combination of lines and circles, and positive and negative data. Some photoplotters have a problem handling the complexity of some custom apertures, so converting the custom aperture to polygonal data helps in this situation.

1. Right-click on the desired custom aperture in the Custom Apertures area. A shortcut menu appears.
2. Select the Flatten command, and the custom aperture is immediately flattened. The D-code association and instances of the custom apertures within your design remain as they were, but their "internal structure" becomes that of polygons.

To copy a custom aperture

1. Right-click on the desired custom aperture in the Custom Apertures area. A shortcut menu appears.
2. Select the Copy command, and a copy of the selected custom aperture now appears in the list. The prefix "Copy_of" is added to the name of the custom aperture you copied. You can now edit and rename the new custom aperture.

To rename a custom aperture

1. Right-click on the desired custom aperture in the Custom Apertures area. A shortcut menu appears.
2. Select the Rename command, and a box appears around the name.
3. Type the new name, and press the Enter key when you are finished.

To view unused custom apertures
1. Right-click in the Custom Apertures heading. A shortcut menu appears.
2. If the Show Unused command has a checkmark next to it, unused apertures are being displayed. If Show Unused does not have a checkmark, select the command to display unused custom apertures in the list.

To delete an unused custom aperture
1. Make sure that unused apertures are displayed (see above).
2. Right-click on the desired custom aperture in the Custom Apertures area. A shortcut menu appears.
3. Select the Delete command, and the custom aperture is immediately deleted.

NC Tools

To see a list of defined NC tool tables, click on the plus box next to the NC Tools heading. The NC Tools area expands to show all the defined tool tables. Clicking on the plus box next to a Tool Table heading reveals all the tools the table contains, with their corresponding tool number and size. Each tool's type is also indicated by a unique symbol.



Most of the basic NC tool information may be modified using the Navigator. If you do not see a procedure here for accomplishing your desired task (such as deleting a tool or changing its color in the workspace), right-click on the NC Tools heading, and select the Spreadsheet command from the shortcut menu. This opens the NC Tool Setup dialog box, where you can access all tool table data.

To add a new tool table
1. Right-click on the NC Tools heading. A shortcut menu appears.
2. Select the Add New Table command. The New Table Name dialog box appears.
3. Specify a name for the new NC tool table, and click OK. The new table is added, and is automatically assigned a type of Both (to accept both Drill and Mill tools).

To change a tool table's name
1. Right-click on the name of the desired tool table. A shortcut menu appears.
2. Select the Rename Table command. A box appears around the table name.
3. Type the new name for the table, and press the Enter key.

To change a tool table's type
1. Right-click on the desired tool table. A shortcut menu appears.
2. Select Change Table Type, and the desired table type (the current type has a check mark next to it).

To add a tool
1. Right-click on the name of the tool table where you wish to add the tool. A shortcut menu appears.
2. Select the Add New Tool command, and the New Tool dialog box appears.
3. By default, the lowest available tool number appears in the text box. Change this to another unused tool number, if desired.
4. Click OK. The tool is assigned default values for size, type, plating, and legend, which you can change in the Navigator.

To make a tool active
1. Right-click on the desired tool in the list. A shortcut menu appears.
2. Select the Active command (when a tool is active, a checkmark appears next to the Active command). The tool is now active.

To change a tool’s size
1. Click on the plus box next to the desired tool.
2. In the tool detail list, the first item is the tool size. Right-click on the size, and a shortcut menu appears.
3. Select the Edit command. A box appears around the size.
4. Type in the new size, and press the Enter key.

To change a tool's type
1. Click on the plus box next to the desired tool.

2. In the tool detail list, the second item is the tool type. Right-click on the type, and a shortcut menu appears.
3. Select the desired tool type from the menu.

To change a tool's plating status

1. Click on the plus box next to the desired tool.
2. In the tool detail list, the third item is the plating status. Right-click on the type, and a shortcut menu appears.
3. A checkmark appears next to the current type. Select either Plated, NonPlated, or Both, as desired.

To change a tool's legend

1. Click on the plus box next to the desired tool.
2. In the tool detail list, the last item is the legend. The legend is one or two alphanumeric characters that are used to designate the tool in a drill hole chart. Right-click on the legend, and a shortcut menu appears.
3. Select the Edit command. A box appears around the legend.
4. Type one or two alpha-numeric characters, and press the Enter key.

To view unused tools

1. To view all unused tools in all tool tables, right-click on the NC Tools heading. A shortcut menu appears.

   -or-

   To view all unused tools in a specific tool table, right-click on the desired tool table name. A shortcut menu appears.
2. Select the Show Unused command, and all unused tools immediately appear in the list.

   To hide unused tools, repeat the above steps. A checkmark now appears next to the Show Unused command. Select the command to remove the checkmark and hide the unused tools.

Nets

The number of nets in your design are noted next to the "Net" heading, in parentheses. To see a list of all nets in your design, click on the plus box next to the Nets heading. The Nets area expands to show all nets, by number.

Tip: To save processing time when you generate a netlist, the Nets list in the Navigator is not populated until you first attempt to access it.

To generate a netlist

1. Right-click on the Nets heading, and a shortcut menu appears.
2. Select the Generate command. The Netlist Generation dialog box appears.
3. Select the desired options and click OK to generate the netlist. After the netlist is generated, all nets are listed under the Nets heading.

To clear an existing netlist

1. Right-click on the Nets heading, and a shortcut menu appears.
2. Select the Clear Existing Netlist command. You are asked to confirm if you want to clear the netlist; doing so can not be undone by selecting the Undo command afterwards.

To query net information

- Double-click on a net name. The Query Net dialog box appears, with detailed information about the selected net.

To highlight a net in the workspace

- Right-click on a net name, and select Highlight from the shortcut menu.

To view/highlight end points in a net

1. Click on the plus box next to the desired net name. A detailed list appears, with the layer number, X and Y coordinates, and User Data for each point. If you have non-contiguous lines in your net, the same coordinates may be listed several times. If you do not have User Data assigned to the net, two successive quotation marks appear next to the X:Y coordinates.
2. To highlight an end point, right-click on the desired coordinates, and select the Highlight command from the shortcut menu. The workspace zooms to the desired coordinates, and the end point is highlighted.

To highlight items not part of a net

In the Navigator, net "None" is a collection of database items that are not part of any net.

1. Right-click on the Nets heading, and a shortcut menu appears.
2. Select the Highlight Net <None> command.

   -or-

- Under the Nets heading, all the existing nets are listed alphanumerically by name. The item in the list, however, is "<None>". Right-click on <None>, and select the Highlight command from the shortcut menu.

To change a net name

1. Right-click on a net name, and a shortcut menu appears.
2. Select the Rename command. A box appears around the text, indicating it can be edited.
3. Type in the new name, and press the Enter key.

To apply external net names

After you have performed a Netlist Comparison, and have corrected any errors, you can use the Apply External Net Names command to apply the net names from the IPC-D-356 netlist to the GerbTool netlist information. The most important purpose for this is that, when you view the net information in the Navigator or query a net, you will see the net name rather than the arbitrary GerbTool net number. If you save your data as a .gtd file, the netlist information is saved as well, so you will no longer need to refer to the external netlist file when viewing your data.

1. Right-click on the Nets heading, and a shortcut menu appears.
2. Select the Apply External Net Names command. The external net names are immediately applied to the nets.

To change the User Data for a net

1. Click on the plus box next to the desired net coordinates. If there is no User Data assigned to a net, "UD <blank>" appears.
2. Right-click on the User Data, and a shortcut menu appears.
3. Select the Edit command. A box appears around the text, indicating it can be edited.
4. Type in the new User Data, and press the Enter key. The user data now also appears next to the net coordinates.

External Nets

When you have imported a netlist, VisualCAM lists the nets in the External Nets area of the Navigator. The number of nets is noted next to the heading. To see a list of all external nets, click on the plus box next to the External Nets heading. The External Nets area expands to show all nets, by net name.

To highlight all the points on an external net

- Double-click on the name of the external net you wish to highlight. All the points for that external net are highlighted in the workspace. To turn off the highlighting, double-click on the external net again.

  -or-

1. Right-click on the name of the net you wish to highlight. A shortcut menu appears.
2. Select the Highlight command. All the points for that external net are highlighted in the workspace. To turn off the highlighting, simply select the Highlight command again (it is a toggle).

To highlight a single external net point

1. Click on the plus box next to the name of the net that contains the desired point. The net points are listed by type. You can view more information about each point by clicking on the plus box net to it.
2. Double-click on the desired external net point. Its location is highlighted in the workspace. To turn off the highlighting, double-click on the external net point in the Navigator again.

   -or-

2. Right-click on the desired external net point, and a shortcut menu appears.
3. Select the Highlight command. The external net location is highlighted in the workspace. To turn off the highlighting, select the Highlight command again (it is a toggle).

Composites

To see a list of all the composites in your design, click the plus box next to the Composites heading.

All composite information may be modified using the Navigator.

To add a new composite

1. Right-click on the Composites heading. A shortcut menu appears.
2. Select the Add Composites command. A new composite with a system-default name appears in the Composites list.

To delete a composite

1. Right-click on the desired composite in the list. A shortcut menu appears.

2. Select the Remove command, and the composite is immediately deleted. This does not delete the actual layers in the database—just the composite information. To delete layers, see the Layers area of the Navigator.

   -or-

1. To delete an empty composite, click on the desired composite in the list.
2. Press the Delete key, and the empty composite is immediately deleted.

Tip: If you delete a composite layer set, the layers will remain tagged as "Composite". You will need to tag your layers with a new layer type.

To add a layer to a composite

1. Click on a layer in the Layers list.
2. While still holding down your mouse button, move the cursor to the desired composite, and release the button (i.e. drag-and-drop the layer into the composite).

   Note that if a layer already resides in a composite, it cannot also be added to another composite.

To remove a layer from a composite

1. Click on the plus box next to the desired composite. A list of layers in the composite appears.
2. Right-click on the desired layer, and a shortcut menu appears.
3. Select the Remove command, and the layer is removed from the composite. This does not delete the actual layer in the database—it just removes the layer from the composite. To delete layers, see the Layers area of the Navigator.

To convert a composite to a single layer

This function converts the composite layers to a single, standard layer. It does not delete the original composite information, or the layers contained in the composite.

- Click on the desired composite, and drag it to the desired empty layer in the Layers list. The composite is immediately converted to a single layer.

  -or-

1. Right-click on the desired composite. A shortcut menu appears.
2. Select the Convert To Layer command. The composite is immediately converted to a single layer, and placed on the next available empty layer.

To hide a composite in the workspace

1. Right-click on the desired composite. A shortcut menu appears, with a checkmark next to the Visible command to indicate the layers are visible.
2. Select the Visible command, and the layers in the composite are hidden from view in the workspace.

   To make the layers visible, select the Visible command again.

Layer Sets

To see a list of all the layer sets in your design, click the plus box next to the Layer Sets heading. The Layer Sets area expands to show all layer sets, by name.

There are three types of layer sets, and each type has its own symbol:

Standard Layer Set
Blind/Buried Layer Set
MCM/LTCC Stackup

Layer sets may be added, removed, modified, or have their view toggled using the Navigator.

To add a new standard or blind/buried layer set

1. Right-click on the Layer Sets heading. A shortcut menu appears.
2. Select Add Layer Set, and the desired type: View/Edit for a Standard Layer Set, or Blind/Buried.
3. A new layer set, with a system-assigned name, is added to the layer set list. You may now add layers to the layer sets (see procedure below).

To add a layer to a standard or blind/buried layer set

1. Click on a layer in the Navigator's Layers list.
2. While still holding down your mouse button, move the cursor to the desired layer set, and release the button (i.e. drag-and-drop the layer into the layer set).

To remove a layer from a standard or blind/buried layer set

1. Click on the plus box next to the desired layer set.
2. A list of the layers in the set appears. Right-click on the desired layer, and a shortcut menu appears.
3. Select the Remove command, and the layer is removed from the set. This does not delete the actual layer in the database—it just removes the layer from the set. To delete layers, see the Layers area of the Navigator.

To add a new MCM/LTCC layer set

1. Right-click on the Layer Sets heading. A shortcut menu appears.
2. Select the Add Layer Set|MCM/LTCC command. The MCM/LTCC Setup dialog box appears.
3. The system looks at all conductive, NC, or insulator layers and automatically attempts to stack the layers. The automatic ordering will be the first conductive layer found, then the first NC/insulator layer found, then the next conductive layer, and so on, until no more NC/insulator or conductive layers are found.
4. To change the layer order, click on the incorrect layer, then select the correct layer from the drop-down list.

   If you need to clear your choices and start the ordering over again, click the Clear button (which then turns into the Reset button), and click the Reset button. The order is set to the system default.
5. Click OK when you are finished. A layer set called "MCM/LTCC_Stackup" appears in the Layer Sets list.

To modify an MCM/LTCC layer set

1. Right-click on the MCM/LTCC layer set in the Navigator. A shortcut menu appears.
2. Select the Edit Stackup command. The MCM/LTCC Setup dialog box appears.
3. To change the layer order, click on the incorrect layer, then select the correct layer from the drop-down list.

   If you need to clear your choices and start the ordering over again, click the Clear button (which then turns into the Reset button), and click the Reset button. The order is set to the system default.
4. Click OK when you are finished.

To rename a layer set

1. Right-click on the desired layer set in the list. A shortcut menu appears.
2. Select the Rename command. An edit box appears around the layer set name.
3. Type in the new name and press the Enter key.

To remove a layer set

1. Right-click on the desired layer set in the list. A shortcut menu appears.
2. Select the Remove command, and the layer set is immediately removed. This does not delete the actual layers in the database—just the layer set information. To delete layers, see the Layers area of the Navigator.

To hide a layer set in the workspace

1. Right-click on the desired layer set. A shortcut menu appears, with a checkmark next to the Visible command to indicate the layers are visible.
2. Select the Visible command, and the layers in the layer set are hidden from view in the workspace.

   To make the layers visible, select the Visible command again.

Assembly

The Assembly area allows you to reverse engineer part information for assembly, as well as view information about your parts and fiducials. Essentially, you can perform all of the functions that are available in the Assembly menu, from the shortcut menus available in the Assembly area of the Navigator.

Click on the + box next to the Assembly heading to access functions specific to Parts and Fiducials.

To view the Footprint Library

The Footprint Library contains information on all part footprints and devices defined for the current database. For more information about the Footprint Library, see *Assembly|Setup Footprint Library*.

1. Right-click on the Assembly heading. A shortcut menu appears.
2. Select the Setup Footprint Library command. The Footprint Library appears.

To import a Bill of Materials (BOM)

After you have reverse engineered your data for assembly, this function allows you to add additional part information from a bill of materials to the database prior to exporting.

1. Right-click on the Assembly heading. A shortcut menu appears

2. Select the Import BOM command. See *Assembly|Import BOM* for further details.

To export a FATF file
After you have finished reverse engineering your data, this function exports a FATF file for use by assembly software.

1. Right-click on the Assembly heading. A shortcut menu appears.
2. Select the Export FATF command. See *Assembly|Export FATF* for further details.

Parts

To see a list of parts that have been identified, click on the plus box next to the Parts heading (if no parts have been identified, the plus box does not appear). The Parts area expands to show all parts. Modifications to individual part instances can be accomplished here. You can also delete footprints or devices.

To identify parts
There are several methods that can be used to reverse engineer Part information.

1. Right-click on the Parts heading. A shortcut menu appears.
2. To identify parts using a part footprint library, select the Identify Using|Footprint Library command. See *Assembly|Parts|Identify Using|Footprint Library* for further details.

   -or-

   To identify parts using an IPC-D-356, select the Identify Using|IPC-D-356 Data command. See *Assembly|Parts|Identify Using|IPC-D-356 Data* for further details.

   -or-

   To identify parts using silkscreen data, select the Identify Using|Silkscreen Layer command. See *Assembly|Parts|Identify Using|SilkScreen Layer* for further details.

   -or-

   To identify parts using centroid data, select the Identify Using|Centroid File command. See *Assembly|Parts|Identify Using|Centroid File* for further details.

   -or-

   To identify parts by manually identifying footprints, select the Identify Using|Select Footprint command. See *Assembly|Parts|Identify Using|Select Footprint* for further details.

To generate a Bill of Materials (BOM) report
This function creates a part report that includes bill of material and centroid information.

1. Right-click on the Assembly heading. A shortcut menu appears.
2. Select the BOM Report command. See *Documentation|Reports|BOM* for further details.

   -or-

1. Click on the + box next to the Assembly heading. Two "branches" appear: Parts and Fiducials.
2. Right-click on the Parts heading. A shortcut menu appears.
3. Select the BOM Report command. See *Documentation|Reports|BOM* for further details.

To sort the Parts list in the Navigator.
If you have parts defined in your database, they are listed under the Parts heading. The default Parts view lists the part instances by reference designator. A checkmark appears next to the current sorting method, in the shortcut menu.

1. Right-click on the Parts heading. A shortcut menu appears.
2. To sort parts by reference designator, select the Sort By|Reference Designator command.

   -or-

   To sort parts by footprint name, select the Sort By|Footprint command.

   -or-

   To sort parts by device name, select the Sort By|Device command

To hide or view part instances in the workspace
If you have parts defined in your database, the data (such as pin numbers, centroids, etc.) exist as "virtual entities". This command is used to toggle the visibility of part data on and off.

1. Right-click on the Parts heading. A shortcut menu appears.
2. The View command is a toggle; a check mark next to it means the view is turned on. Select the command to change the viewing state.

To delete all part instances
This command deletes all part instances from the database. It does not, however, affect the footprint library.

1. Right-click on the Parts heading. A shortcut menu appears.
2. Select the Delete All command. You are prompted to confirm that you wish to delete all parts.

To delete an individual part instance
This command deletes a part instance from the database. It does not, however, affect the footprint library.

1. Right-click on the Parts heading. A shortcut menu appears
2. Select the Delete command. See *Assembly*|*Parts*|*Delete* for further details.

-or-

1. In the Parts list, right-click on the reference designator of the part instance you wish to delete. A shortcut menu appears.
2. Select the Delete command. You are prompted to confirm that you wish to delete the part.

To query a part in the workspace
1. Right-click on the Parts heading. A shortcut menu appears.
2. Select the Query command. Click on the part in the workspace that you wish to view information for. See *Assembly*|*Parts*|*Query* for further details.

To highlight a part in the workspace
1. In the Parts list, right-click on the reference designator for the desired part. A shortcut menu appears.
2. Select the Highlight command. The selected part instance is highlighted in the workspace.

To highlight all instances of a particular device in the workspace
1. Sort the Parts list by Device.
2. Right-click on the desired device name. A shortcut menu appears.
3. Select the Highlight command. The selected device instances are highlighted in the workspace.

To highlight all instances a particular footprint in the workspace
1. Sort the Parts list by Footprint.
2. Right-click on the desired footprint. A shortcut menu appears.
3. Select the Highlight command. All parts that use that footprint are highlighted in the workspace.

To change a reference designator
1. In the Parts list, right-click on the reference designator you wish to change. A shortcut menu appears.
2. Select the Rename command. An edit box appears around the text.
3. Type the new reference designator, and press the Enter key.

To rotate a part instance
1. In the Parts list, right-click on the reference designator of the part instance you wish to rotate. A shortcut menu appears.
2. Select the Rotate command, and the amount of rotation you wish to apply. Rotation is only available if a part can be rotated, and only viable rotations are offered.

To identify a part fiducial
1. In the Parts list, right-click on the reference designator for the part you wish to identify the fiducial for. A shortcut menu appears.
2. Select the ID Fiducials command. You are prompted to select the fiducial for the part.
3. Click on the fiducial in the workspace. It is tagged as a part fiducial in the workspace (identified with a target) and added to the part fiducials list (in the Fiducials area of the navigator).
4. Press the Esc key when you are finished.

To change the device type for a part
1. In the Parts list, click on the + box next to the reference designator of the desired part. Additional part information appears.
2. Right-click on the part's device type. A list of available device types appears.
3. Select the desired device type.

To delete a device
1. Sort the Parts list by Device.
2. Right-click on the desired device. A shortcut menu appears.

3. Select the Delete command. The Device Replacement dialog box appears.
4. In the Replacement Device list, select the device that you want assigned to the associated part instances. If you select None, all part instances that are associated with the selected device will be deleted from the database, in addition to the device.

To delete a footprint
This deletes the footprint from the library, as well as all the part instances that use it.

1. Sort the Parts list by Footprint.
2. Right-click on the desired footprint name. A shortcut menu appears.
3. Select the Delete command. You are asked to confirm that you wish to delete the footprint, and all the part instances that use it in the design.

Fiducials

To see lists of board and part fiducials that have been identified, click on the plus box next to the Fiducials heading (if no fiducials have been identified, the plus box does not appear). All fiducials are listed by type.

To identify fiducials

1. Right-click on the Fiducials heading. A shortcut menu appears.
1. Select the Identify command. See *Assembly*|*Fiducials*|*Identify* for further details.

To generate a fiducial report
This function creates a report of the names, locations, and types of fiducials on your board.

1. Right-click on the Assembly heading. A shortcut menu appears.
2. Select the Fiducial Report command. See *Documentation*|*Reports*|*Fiducials* for further details.

-or-

1. Click on the + box next to the Assembly heading. Two "branches" appear: Parts and Fiducials.
2. Right-click on the Fiducials heading. A shortcut menu appears.
3. Select the Report command. See *Documentation*|*Reports*|*Fiducials* for further details.

To hide or view all fiducials in the workspace
All fiducials are marked in the workspace with a target.

1. Right-click on the Fiducials heading. A shortcut menu appears.
2. The Highlight command is a toggle; a check mark next to it means the view is turned on. Select the command to change the viewing state.

To hide or view board fiducials in the workspace
All fiducials are marked in the workspace with a target.

1. Right-click on the Board heading. A shortcut menu appears.
2. The Highlight command is a toggle; a check mark next to it means the view is turned on. Select the command to change the viewing state.

-or-

1. To hide/view all top-side board fiducials, click on the + box next to the Board heading. The board fiducials are grouped by the side of the board they are placed on.
2. Right-click on the Top heading. A shortcut menu appears.
3. The Highlight command is a toggle; a check mark next to it means the view is turned on. Select the command to change the viewing state.

-or-

1. To hide/view all bottom-side board fiducials, click on the + box next to the Board heading. The board fiducials are grouped by the side of the board they are placed on.
2. Right-click on the Bottom heading. A shortcut menu appears.
3. The Highlight command is a toggle; a check mark next to it means the view is turned on. Select the command to change the viewing state.

-or-

1. To hide/view an individual fiducial, right-click on the desired fiducial name in the Top or Bottom list. A shortcut menu appears.
2. The Highlight command is a toggle; a check mark next to it means the view is turned on. Select the command to change the viewing state.

To hide or view part fiducials in the workspace

All fiducials are marked in the workspace with a target. They are grouped in the Navigator by the side of the board they are on.

1. Right-click on the Part heading. A shortcut menu appears.
2. The Highlight command is a toggle; a check mark next to it means the view is turned on. Select the command to change the viewing state.

-or-

1. To hide/view all top-side part fiducials, click on the + box next to the Part heading.
2. Right-click on the Top heading. A shortcut menu appears.
3. The Highlight command is a toggle; a check mark next to it means the view is turned on. Select the command to change the viewing state.

-or-

1. To hide/view all bottom-side part fiducials, click on the + box next to the Part heading.
2. Right-click on the Bottom heading. A shortcut menu appears.
3. The Highlight command is a toggle; a check mark next to it means the view is turned on. Select the command to change the viewing state.

-or-

1. To hide/view an individual fiducial, right-click on the desired fiducial name in the Top or Bottom list. A shortcut menu appears.
2. The Highlight command is a toggle; a check mark next to it means the view is turned on. Select the command to change the viewing state.

To rename a board fiducial

1. Right-click on the desired fiducial name in the Board Top or Bottom list. A shortcut menu appears.
2. Select the Rename command. An edit box appears around the text.
3. Type in the new fiducial name, and press the Enter key.

To change the assignment of a part fiducial

1. Right-click on the desired reference designator. A shortcut menu appears.
2. Select the Change Reference Designator command. A menu of reference designators appear.
3. Select the desired reference designator. The fiducial is reassigned to that part instance.

To delete all fiducials

1. Right-click on the Fiducials heading. A shortcut menu appears.
2. Select the Delete All command.

To delete board fiducials

1. To delete all board fiducials, right-click on the Board heading. A shortcut menu appears.
2. Select the Delete command.

-or-

1. To delete all top-side board fiducials, click on the + box next to the Board heading.
2. Right-click on the Top heading. A shortcut menu appears.
3. Select the Delete command.

-or-

1. To delete all bottom-side board fiducials, click on the + box next to the Board heading.
2. Right-click on the Bottom heading. A shortcut menu appears.
3. Select the Delete command.

-or-

1. To delete an individual fiducial, right-click on the desired fiducial name in the Top or Bottom list. A shortcut menu appears.
2. Select the Delete command.

To delete part fiducials

1. To delete all part fiducials, right-click on the Part heading. A shortcut menu appears.
2. Select the Delete command.

-or-

1. To delete all top-side part fiducials, click on the + box next to the Part heading.
2. Right-click on the Top heading. A shortcut menu appears.
3. Select the Delete command.

-or-

1. To delete all bottom-side part fiducials, click on the + box next to the Part heading.
2. Right-click on the Bottom heading. A shortcut menu appears.
3. Select the Delete command.

-or-

1. To delete an individual fiducial, right-click on the desired fiducial name in the Top or Bottom list. A shortcut menu appears.
2. Select the Delete command.

Embedded Passives

When you tag a passive layer as "Passive Add" or "Passive Sub", it is immediately added to the Embedded Passives area of the Navigator. When you have reverse engineered your embedded passive data, you can view information about the embedded passives on those layers, and highlight specific embedded passives in the workspace. To view the detail list for a passive layer, click on the plus (+) box next to the desired passive layer.

The following items appear in the detail list, in this order:

- The conductive layer number and name.
- The material constant for the layer, in Ohms per square inch.
- The Valid Embedded Passives list. A valid embedded passive is a flash on the passive layer which has been defined as a resistor shape, and has two termination bar flashes associated to it on the conductive layer. The embedded passives are listed by reference designator. If you click on the plus box next to the reference designator, the embedded passive's coordinates and resistance value are displayed.
- The Invalid Embedded Passives list. This list contains flashes that have been marked as resistor shapes, but have not been associated with an embedded passive. They are listed by a system-assigned number.
- The Non-Embedded Passive Flashes list. This is a list of all flashes on the passive layer, by their X:Y coordinates, that have not been defined as resistor shapes.

To change the conductive layer

You can only change the conductive layer if no valid embedded passives have been defined for the passive layer.

1. Right-click on the desired conductive layer, and a shortcut menu appears.
2. Select the desired conductive layer from the menu list.

To change the material constant

1. Right-click on the material constant amount, and an edit box appears around it.
2. Type the new amount, and press the Enter key.

To mark flashes as resistor shapes

To mark all flashes

1. Right-click on the Non-Embedded Passive Flashes heading. A shortcut menu appears.
2. Select the Mark All As Embedded Passive command. All flashes are now marked as resistor shapes, and appear in the Invalid Embedded Passives list.

To mark flashes individually

1. Click on the plus box next to the Non-Embedded Passive Flashes heading, and the flashes are listed by X:Y coordinate.
2. Right-click on the desired flash, and a shortcut menu appears.
3. Select the Mark As Embedded Passive command. The selected flash has now been marked as a resistor shape, and appears in the Invalid Embedded Passives list.

To define embedded passives from Non-Embedded Passive Flashes

1. Click on the plus box next to the Non-Embedded Passive Flashes heading, and the flashes are listed by X:Y coordinate.
2. Right-click on the desired flash, and a shortcut menu appears.

3. Select the Construct Embedded Passive command. The selected flash is immediately highlighted in the workspace, and you are prompted to select one of its associated termination bars.
4. Click on the first termination bar. You are prompted to select the second one.
5. Click on the second termination bar. A dialog box appears, prompting you to specify a reference designator for the embedded passive.
6. Type the reference designator, and click the OK button. The embedded passive now appears in the Valid Embedded Passives list.

To define embedded passives from Invalid Embedded Passives

1. Click on the plus box next to the Invalid Embedded Passives heading to view the resistor shapes. If you click on the plus box next to system-assigned number, you can view the coordinates of the resistor shape.
2. Right-click on the system-assigned number, and a shortcut menu appears.
3. Select the Construct Embedded Passive command. The selected resistor shape is immediately highlighted in the workspace, and you are prompted to select one of its associated termination bars.
4. Click on the first termination bar. You are prompted to select the second one.
5. Click on the second termination bar. A dialog box appears, prompting you to specify a reference designator for the embedded passive.
6. Type the reference designator, and click the OK button. The embedded passive now appears in the Valid Embedded Passives list.

To change a reference designator

1. In the Valid Embedded Passives list, right-click on the desired reference designator. A shortcut menu appears.
2. Select the Change Reference Designator command. An edit box appears around the reference designator.
3. Type the new reference designator, and press the Enter key.

To destroy embedded passives

Destroying embedded passives removes the association of the resistor shape and the termination bars.

To destroy all embedded passives

1. Right-click on the Valid Embedded Passives heading. A shortcut menu appears
2. Select the Destroy All command. The link between the resistor shapes and termination bars is broken, but the resistor shapes still remain marked as resistors. They are now listed in the Invalid Embedded Passives list.

To destroy embedded passives individually

1. In the Valid Embedded Passives list, right-click on the desired reference designator. A shortcut menu appears.
2. Select the Make Invalid command. The link between the resistor and termination bars is broken, but the resistor shape still remains marked as a resistor. It is now listed in the Invalid Embedded Passives list.

To destroy resistor shapes

Destroying resistor shapes removes their designation as resistors, and they are considered ordinary flashes.

To destroy all resistor shapes

1. Right-click on the Invalid Embedded Passives heading. A shortcut menu appears.
2. Select the Destroy All command. The resistor shapes are now designated as ordinary flashes, and listed in the Non-Embedded Passive Flashes list.

To destroy resistor shapes individually

1. In the Invalid Embedded Passives list, right-click on the reference designator/number of the desired passive shape. A shortcut menu appears.
2. Select the Destroy command. The resistor shape is now designated as an ordinary flash, and listed in the Non-Embedded Passive Flashes list.

To hide the Ohms symbol

Every flash that is tagged as a resistor on the passive layer is marked with an Ohms symbol.



Resistor Flash

1. To hide the Ohms symbol on resistor shapes, right-click on the Embedded Passives heading. A shortcut menu appears. A check mark next to the Show Ohms Symbol command indicates the symbols are shown.
2. Select the Show Ohms Symbol command. The Ohms symbol no longer appears on resistor flashes in the workspace. To make the symbol appear again, select the Show Ohms Symbol command again.

To hide the outlines around embedded passives

Every embedded passive that has been built is highlighted with an outline.



"Built" Embedded Passive

1. To hide the outlines around embedded passives, right-click on the Embedded Passives heading. A shortcut menu appears. A check mark next to the Show Outlines command indicates that outlines are shown.
2. Select the Show Outlines command. The outlines no longer appear around embedded passives in the workspace. To make the outlines appear again, select the Show Outlines command again.

To highlight all valid embedded passives

1. Right-click on the Valid Embedded Passives heading. A shortcut menu appears.
2. Select the Highlight command, and all the valid embedded passives for that passive layer are highlighted in the workspace.

To highlight all invalid embedded passives

1. Right-click on the Invalid Embedded Passives heading. A shortcut menu appears.
2. Select the Highlight command, and all resistor shapes that are not part of a valid embedded passive are highlighted in the workspace.

To highlight all non-embedded passive flashes

1. Right-click on the Non-Embedded Passive Flashes heading. A shortcut menu appears.
2. Select the Highlight command, and all flashes that have not been marked as resistor shapes are highlighted in the workspace.

To zoom to a valid embedded passive

1. Click on the plus box next to the Valid Embedded Passives heading. A list of embedded passives appears, sorted by reference designator.
2. Right-click on the desired reference designator. A shortcut menu appears.
3. Select the Zoom To command. The workspace immediately zooms to the desired embedded passive.

To zoom to an invalid embedded passive

1. Click on the plus box next to the Invalid Embedded Passive Flashes heading. A list of resistor shapes appears, by system-assigned number.
2. Right-click on the number. The workspace immediately zooms to the desired resistor shape.

To zoom to a non-embedded passive flash

1. Click on the plus box next to the Non-Embedded Passive Flashes heading. A list of flashes appears, by coordinate.
2. Right-click on the desired coordinate. The workspace immediately zooms to the desired flash.

Analysis

When you run a DRC/MRC analysis, Netlist Comparison, Layer Comparison, and Duplicate Data analysis of your design, generate/optimize a solder mask or paste mask layer, or reverse engineer data for assembly, the errors are logged in the Analysis heading. For DRC/MRC analyses and netlist comparisons, each run that is performed is logged using the assigned Run Name. If you do not assign a Run Name, the run is identified by the time and date it is performed. Each pass that is performed in the run is listed, and is categorized by the type of analysis (DRC, DFF, etc.). Duplicate Data runs are logged under "Duplicate Data", and Layer Comparisons are logged under "Compare Layers" with the layer numbers, by default. Solder and Paste mask errors are logged under the appropriate layer type ("Top Solder Mask", "Bot Solder Mask", etc.).

Viewing Analysis Settings & Errors

To view information about a run

1. Right-click on the desired run. A shortcut menu appears.

2. Select the Properties command. The Analysis Properties dialog box appears.

   The run name, start and end times are given, along with the settings you specified prior to starting the run.

3. Click the Close button when you are finished.

To view information about a pass

1. Right-click on the desired pass in the list. A shortcut menu appears.
2. Select the Properties command. The Analysis Properties dialog box appears, with the Pass tab selected.

   The pass name, drill layer number, and the conductive layers you chose to analyze are indicated. The Analysis Type list reflects the settings you selected prior to starting the run. If you would like to view the run settings, select the Run tab.

3. Click the Close button when you are finished.

To view information about an error

1. Right-click on the desired error in the list. A shortcut menu appears.
2. Select the Properties command. The Analysis Properties dialog box appears, with the Error tab selected.

   The error type and system-assigned ID number are provided. The Fix State tells you if the error can be automatically fixed by VisualCAM. The rule you specified prior to the run is indicated, as well as the properties of the error that violated the rule. Additional aperture, layer, coordinate, and net information are also provided.

   If you would like to view the run or pass settings, click on the appropriate tab. If you would like to view information about another error, click on it in the error list (you do not have to close the dialog box).

Tip: If you want to keep track of certain individual errors, make a note of the ID number. You can use it to search for the error later (see the "To search for an error" procedure below).

3. Click the Close button when you are finished.

To search for an error

1. Right-click on the Analysis heading. A shortcut menu appears.
2. Select the Find Error command.
3. Type the error ID number in the dialog box (see the "To view information about an error" procedure above, if you do not know the error's ID) and click OK. The error is selected in the Navigator list.

To highlight errors in the workspace

When an error is highlighted, the light bulb icon next to it is "lit" (yellow). If an error cannot be highlighted, the round "X" icon next to the error is black.

1. To highlight all the errors that were found, right-click on the error category. A shortcut menu appears.
2. Select the Highlight All command. The workspace zooms to the extent of the items that are highlighted.

   -or-

- To highlight a single error, double-click on it in the error list. The workspace zooms to the selection.

  If you would prefer to use a single mouse click to highlight an error, right-click on the Analysis heading. A shortcut menu appears. Select the Single Click Highlight command. If you wish to return to double-click highlighting, select the Single Click Highlight command again to remove its checkmark.

  -or-

1. To highlight multiple errors of your choice, press and hold down the Ctrl key, and click on all the desired errors in the error list. Release the Ctrl key when you are finished. If you wish to highlight several sequential errors in the list, press and hold down the Shift key, then select the first and last error you wish to highlight. Release the Shift key when you are finished.
2. Right-click on the error list. A shortcut menu appears. Select the Highlight command. The workspace zooms to the extent of the items that are highlighted.

To turn off the highlighting of errors in the workspace

When an error is not highlighted, the light bulb next to it is "off" (white).

1. To turn off the highlighting for all errors, right-click on the analysis type. A shortcut menu appears.
2. Select the Unhighlight All command. The highlighting in the workspace is removed.

   -or-

- To turn off the highlighting of a single error, double-click on the highlighted error in the list.

  If you would prefer to use a single mouse click to remove highlighting, right-click on the Analysis heading. A shortcut menu appears. Select the Single Click Highlight command. If you wish to return to double-click

highlighting, select the Single Click Highlight command again to remove its checkmark. The highlighting in the workspace is removed.

-or-

1. To turn off the highlighting of multiple errors, press and hold down the Ctrl key, and click on all the desired errors in the error list. Release the Ctrl key when you are finished. If you wish to remove highlighting for several sequential errors in the list, press and hold down the Shift key, then select the first and last error in the group. Release the Shift key when you are finished.
2. Right-click on the error list. A shortcut menu appears. Select the Highlight command to remove the checkmark. The highlighting in the workspace is removed.

To find the location of an open or short in the workspace

1. Right-click on the Open or Short in the error list. A shortcut menu appears.
2. Select the Pin-Point Error command. The system zooms to the location of the error, and places a bounding box around the area where the open or short is located.

To hide fixed errors in the Navigator

1. Right-click on the Analysis heading. A shortcut menu appears.
2. Select the Show Fixed command to remove the checkmark. All fixed errors (those marked with a green "X" icon) are hidden from view. If you wish to view these errors again, select the Show Fixed command again.

Fixing Errors

To automatically fix errors

To view the results of the fixes, we recommend that you highlight them before you fix them. Note that VisualCAM may not be able to automatically fix some errors, and cannot fix netlist or layer comparison problems. You will have to modify the data for those errors yourself.

1. To fix an individual error, right-click on it. A shortcut menu appears.
2. Select the Fix command. When the error is fixed, the round "X" icon next to the error turns green.

   For DRC/MRC violations, the database item is altered so that it no longer violates your DRC/MRC rules.

   For duplicate data, the offending item is deleted.

   For solder mask errors, the following fixes are applied:

   Missing Electrical Pad
   There is no pad on the electrical layer corresponding to an opening on the mask layer. To fix this, the mask opening is deleted.

   Missing Mask Opening
   There is no opening on the mask layer corresponding to a pad on the electrical layer. To fix this, a mask opening is added.

   Misalignment Error
   The opening on the mask layer and its corresponding pad are misaligned by a value greater than the Location Alignment value you specified. To fix this, the mask opening is moved to the same location as the pad.

   Mask Opening Minimum Oversize Error
   The mask opening is not oversized by at least the minimum oversize specified. To fix this, the mask opening is enlarged to the minimum oversize amount.

   Mask Opening Spacing Error
   Two mask openings violate the minimum distance between mask openings. This might create mask slivers. To fix this, the openings are shaved or reduced (as specified by you).

   Mask Opening To Trace Error
   The mask opening is too close to a trace and violates the minimum mask to trace distance specified. The trace might be exposed in the mask opening. To fix this, the openings are shaved or reduced (as specified by you).

   For paste mask errors, the following fixes are applied:

   Missing Electrical Pad
   There is no pad on the electrical layer corresponding to an opening on the mask layer. To fix this, the mask opening is deleted.

   Missing Mask Opening
   There is no opening on the mask layer corresponding to a pad on the electrical layer. To fix this, a mask opening is added.

Misalignment Error
The opening on the mask layer and its corresponding pad are misaligned by a value greater than the Location Alignment value you specified. To fix this, the mask opening is moved to the same location as the pad.

Mask Opening Minimum Undersize Error
The mask opening is not undersized by at least the minimum specified undersize. To fix this, the mask opening is undersized by the specified amount.

For embedded passive errors, the following fixes are applied:

EP Alignment
The resistor shape is misaligned. To fix this, the resistor shape is moved so that it is centered between the termination bars.

EP TBar Overlap
The resistor shape does not overlap the termination bars enough. To fix this, the resistor shape is altered so that the minimum overlap is attained.

EP TBar Extension
The termination bar is not the correct length. For additive passives, the length of the termination bar is increased to attain the minimum extension length. You specify the minimum copper clearance that must be maintained when the fix is made.

For subtractive passives, the length of the termination bar is decreased to attain the minimum extension of the resistor shape beyond the ends of the termination bars.

EP TBar Width
The termination bar is not wide enough. To fix this, the width of the termination bar is increased so that it meets the minimum specified width. This can be done one of three ways:

- **Uniformly** The width is extended equidistantly on the both sides of the termination bar. You specify the minimum copper clearance that must be maintained when the fix is made.
- **Inside** The width is extended only towards the resistor.
- **Outside** The width is extended away from the resistor. You specify the minimum copper clearance that must be maintained when the fix is made.

EP Minimum Size
The active resistor area is too small. No automatic fix is provided for this.

EP Maximum Size
The active resistor area is too large. No automatic fix is provided for this.

EP To Copper
The resistor shape is too close to adjacent copper. No automatic fix is provided for this.

EP To Drill
The resistor shape is too close to drills on the same net. No automatic fix is provided for this.

For test point analysis errors, the following fixes are applied:

Missing Test Point
There are less than two test points for a given net and there are at least two flashes (pads) on the net. Fixing this error will add a test point to each pad on the net that does not already have a test point. If solder mask accessibility was specified in the Test Point Setup, the test point will only be added if the flash is accessible.

Too Few Flashes
There are less than two test points for a given net and there are less than two pads in the net. No automatic fix is provided for this.

-or-

1. To fix multiple errors, press and hold down the Ctrl key, and click on all the desired errors in the error list. Release the Ctrl key when you are finished. If you wish to fix several sequential errors in the list, press and hold down the Shift key, then select the first and last error in the group. Release the Shift key when you are finished.
2. Right-click on the error list. A shortcut menu appears. Select the Fix command. For DRC/MRC violations, the database item is altered so that it no longer violates your DRC/MRC rules. For duplicate data, the offending item is deleted. The round "X" icon next to the error turns green.

   If there are any errors in your selection group that cannot be fixed, the Fix command will not be available in the shortcut menu.

   -or-

1. To fix all errors, right-click on the analysis type. A shortcut menu appears.
2. Select the Fix All command. For DRC/MRC violations, the database item is altered so that it no longer violates your DRC/MRC rules. For duplicate data, the offending item is deleted. The round "X" icon next to the error turns green. The icons next to any errors that could not be automatically fixed remain red.

You can reverse a fix by right-clicking on the error again, and selecting the Unfix command from the shortcut menu. The data are returned to their original state.

⚠ *Warning: If you save your database, the fix memory buffer is cleared and you will no longer be able to "unfix" an error.*

If you want to run the analyses again after you have fixed the errors (to verify no errors still exist), you should first regenerate the netlist. This is because any fixes will not be reflected in the netlist that you created before the initial analyses.

Creating Reports

To generate a summary report file for all runs

1. Right-click on the Analysis heading. A shortcut menu appears.
2. Select the Report command. A Save As dialog box appears.
3. Select the desired folder and specify the report file name. A text file containing error information for all your runs is created.

To generate a summary report file for an individual run

1. Right-click on the desired run. A shortcut menu appears.
2. Select the Report command. A Save As dialog box appears.
3. Select the desired folder and specify the report file name. A text file containing error information for the run is created.

To print reports for all the errors in a run

1. Right-click on the desired run. A shortcut menu appears.
2. Select the Print All command. The reports are sent to your default printer.

   A full-page report is created for each individual error. It contains a picture of the data in question (both zoomed-in and a diagram showing where it is in the design area), the date, file name, units of measure, information about the error and the item that caused the error.

To generate a summary report file for all errors in a pass

1. Right-click on the Analysis heading. A shortcut menu appears.
2. Select the Report command. A Save As dialog box appears.
3. Select the desired folder and specify the report file name. A text file containing information for all your errors is created.

To print reports for all the errors in a pass

1. Right-click on the desired pass. A shortcut menu appears.
2. Select the Print All command. The reports are sent to your default printer.

   A full-page report is created for each individual error. It contains a picture of the data in question (both zoomed-in and a diagram showing where it is in the design area), the date, file name, units of measure, information about the error and the item that caused the error.

To generate a summary report file for all errors of a particular type

1. Right-click on the desired error category (such as "DRC"). A shortcut menu appears.
2. Select the Report command. A Save As dialog box appears.
3. Select the desired folder and specify the report file name. A text file containing information for all your errors is created.

To print reports for all the errors of a particular type

1. Right-click on the desired error category (such as "DRC"). A shortcut menu appears.
2. Select the Print All command. The reports are sent to your default printer.

   A full-page report is created for each individual error. It contains a picture of the data in question (both zoomed-in and a diagram showing where it is in the design area), the date, file name, units of measure, information about the error and the item that caused the error.

To generate a summary report file for an individual error

1. Right-click on the desired error. A shortcut menu appears.
2. Select the Report command. A Save As dialog box appears.

3. Select the desired folder and specify the report file name. A text file containing information for the error is created.

To print a report for an individual error

1. Right-click on the desired error. A shortcut menu appears.
2. Select the Print command. The report is sent to your default printer.

   A full-page report is created for the error. It contains a picture of the data in question (both zoomed-in and a diagram showing where it is in the design area), the date, file name, units of measure, information about the error and the item that caused the error.

Editing & Deleting Run Information

To rename a run

1. Right-click on the run name. A shortcut menu appears.
2. Select the Rename command. A box appears around the name, indicating that it can be edited.
3. Type in the new run name, and press the Enter key when you are finished.

To delete a run

1. Right-click on the desired run. A shortcut menu appears.
2. Select the Delete command. The run information is deleted from the Navigator.

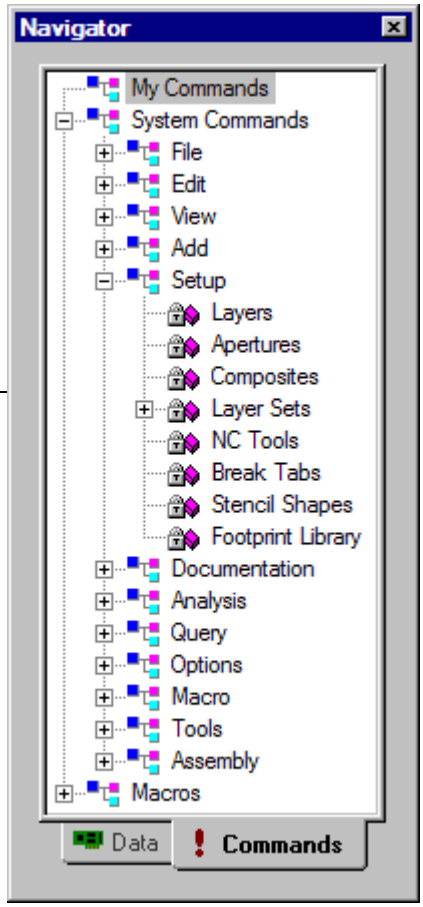
To delete all runs

1. Right-click on the Analysis heading. A shortcut menu appears.
2. Select the Delete All command. The information about all runs is deleted from the Navigator.

Related topics

Edit|Align Layers
Tools|Fix SilkScreen

Commands Tab



The Commands tab provides a shortcut to the main menu commands and macros. The information is displayed in a tree format, with expandable/collapsible lists. By default, the My Commands area contains no functions, the System Commands are structured as they are in the main menu, and the Macros area contains all loaded macros. To expand an area of the tree, click on the plus (+) box next to the desired database element. The "branches" of the information hierarchy are shown, and the plus box becomes a minus (-) box. To compress or hide the information, click on the minus box. If there is no Plus symbol next to a topic in the tree, that area contains no information.

The state of the Commands Tab remains the same regardless of what database is loaded.

My Commands

If you frequently use certain commands or macros, you can place shortcuts to the commands/macros in the My Commands area. The My Commands list may be sorted by clicking and dragging the commands/macros up or down in the list. The My Commands area can help speed up your processes and provide an organized list of steps that must be taken for your jobs.

Tip: Adding, deleting, or renaming a command or macro in the My Commands list has no affect on the command or macro itself. You are only modifying the shortcuts to these functions.

To rename the My Commands list

1. Right-click on the My Commands heading. A shortcut menu appears.
2. Select the Rename command. An edit box appears around the heading.
3. Type in the desired name, and press the Enter key.

To add branches to the My Commands area

1. Right-click on the My Commands heading. A shortcut menu appears.
2. Select the Add Branch command. An empty branch with a system-assigned name appears. You can now click and drag commands/macros to this branch.

   If desired, you can also add "sub-branches" by right-clicking on a branch and selecting the Add Branch command.

To rename branches

1. Right-click on the desired branch name. A shortcut menu appears.
2. Select the Rename command. An edit box appears around the name.
3. Type in the desired name, and press the Enter key.

To delete branches

1. Right-click on the desired branch. A shortcut menu appears.
2. Select the Delete command. The branch, and any command/macro shortcuts within it, is deleted.

   -or-

1. Click on the desired branch.
2. Press the Delete key. The branch, and any command/macro shortcuts within it, is deleted.

To add menu commands to the My Commands list

1. Click on the plus box next to the System Commands heading. A list of all menus is provided.
2. Click on the plus box next to the desired menu (and sub-menu, if necessary) until you see the command you wish to put in the My Commands list.
3. Click on the command and hold down the mouse button.
4. Drag the cursor to the desired location in the My Commands area, and release the button. The command now appears in the list. (The command/macro still appears in the System Commands or Macros area. You are simply creating a shortcut.)

To add macros to the My Commands list

1. Click on the plus box next to the Macros heading. A list of all macros is provided.
2. Click on the desired macro and hold down the mouse button.
3. Drag the cursor to the My Commands area, and release the button. The macro now appears in the list.

To execute a command/macro

- Double-click on the desired command/macro. It executes immediately.

To rename a command/macro

1. Right-click on the desired command or macro. A shortcut menu appears.
2. Select the Rename command. An edit box appears around the name.
3. Type in the desired name, and press the Enter key.

To delete a command/macro

1. Right-click on the desired command/macro. A shortcut menu appears.
2. Select the Delete command.

   -or-

1. Click on the desired command/macro.
2. Press the Delete key.

System Commands

The System Commands area provides another way to execute VisualCAM menu commands. To see a list of all the VisualCAM menus, click the plus box next to the System Commands heading. The System Commands area expands to show all menus, in the order they appear in the VisualCAM main window. You cannot rename commands or reorganize the lists.

To execute a command

1. Click on the plus box next to the desired menu heading. A list of commands for that menu appears, as they are listed in the actual menu in the VisualCAM main window.
2. Double-click on the desired command. It executes immediately.

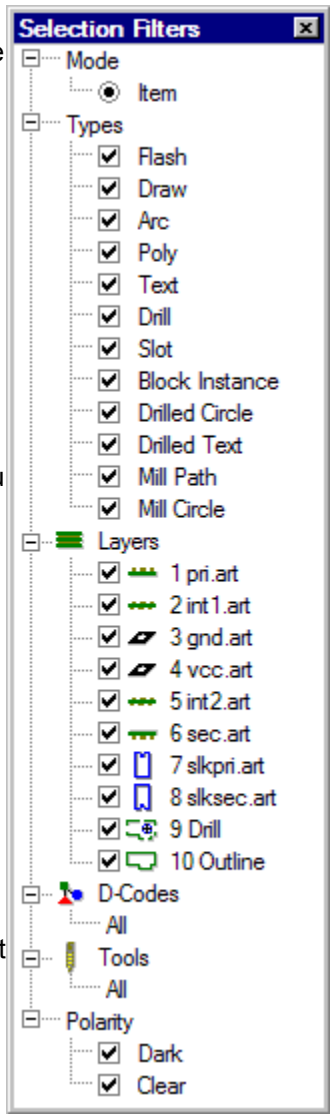
Macros

The Macros area provides another way to execute VisualCAM macros. To see a list of all the VisualCAM macros currently loaded, click the plus box next to the Macros heading. The Macros area expands to show all macros, by name. You cannot rename macros or reorganize the list. To add a macro to the list, use the Macro|Load command.

To execute a macro

- Double-click on the desired macro. It executes immediately.

Selection Filter



The Selection Filter allows you to define how you want to select items, and what types of items you wish to choose from. For example, depending on the command, you may choose from single item, window, group, or complete layer selections, as well as restricting your selections to particular layers and/or D-codes, etc.

When you are not in a command that uses the Selection Filter, the text in the Selection Filter is gray and you cannot select any options. You also do not have the option of using the View|Selection Filter command or the F hotkey when not in a function that uses the Selection Filter.

You can close the display at any time by clicking on the X in the upper-right corner. If you undock the Selection Filter from the VisualCAM window while not in a function that uses it, it will disappear. It will automatically appear again when a function requires it. You can expand the size of the bar to the entire height of the workspace by clicking the up-arrow; then reduce its size by clicking the down-arrow. While it is docked in the VisualCAM window, you can resize the height and width of the bar by moving your cursor to one of its outer edges. When the cursor changes to ⇤⇥, click and hold the left mouse button while you drag the edge to the desired size.

Mode
These options define how items are selected. The available modes depend upon the function you are in. When you are in Item select mode, a bounding box is added to the cursor. Anything that falls within this bounding box is a potential selection. You can increase the size of the box by pressing the PgUp hotkey, or decrease the size (for increased accuracy) by pressing the PgDn hotkey. If more than one item exists at the point you select, the Choose Selection dialog box appears.

In Window select mode, everything that falls *completely* within your selection window is selected. In Window +Xing mode, everything inside the selection window, including anything that the window touches, is selected.

In Layer select mode, all selected types of data on all selected layers are chosen.

Types
These options allow you to select which kinds of items will be included. Any type of item not checked will be excluded from selection.

Layers
These options allow you to select which layers will be considered when choosing items.

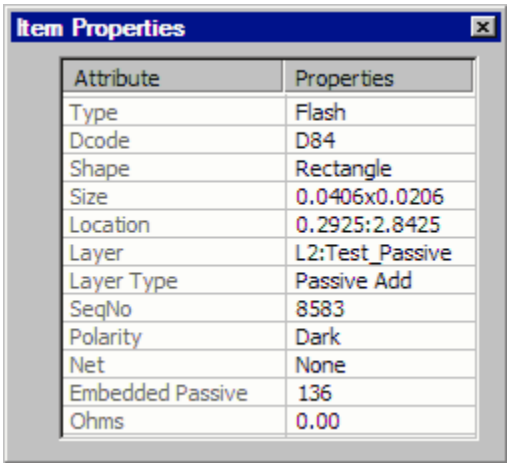
D-Codes
These options allow you to specify items that are created using only a particular D-code.

Tools
These options allow you to specify drill or mill items that are created using a particular NC tool.

Polarity
These options allow you to specify only items with a particular polarity.

Item Properties Display

When you query an item in the database, its information is shown in the Item Properties display. The type of information displayed depends upon the item being queried. The information for each item type is detailed below.



You can close the display by clicking on the X in the upper-right corner. It will automatically appear again when a function requires it. You can expand the size of the bar to the entire height of the workspace by clicking the up-arrow; then reduce its size by clicking the down-arrow. While it is docked in the VisualCAM window, you can resize the height and width of the bar by moving your cursor to one of its outer edges. When the cursor changes to ⇹, click and hold the left mouse button while you drag the edge to the desired size.

Each Item Type that can be queried is listed below, with all the applicable information that can be provided for it.

Arc

CCW indicates the arc is created in a counter-clockwise direction, CW indicates a clockwise direction.

- **Dcode** The D-code used to create the arc.
- **Shape** The shape of the D-code used.
- **Size** The size of the D-code used.
- **From** The X:Y location where the arc begins.
- **To** The X:Y location where the arc ends.
- **Center** The center point of the arc.
- **Length** The length of the arc.
- **Area** The total square area of the arc.
- **Layer** The layer the item is on
- **Layer Type** The type of layer the item is on.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.
- **Polarity** The polarity of the item, either positive or negative.
- **Net** The net number that the item belongs to (if any).

Block Instance

Also known as "panel images".

- **FilePath** The name of the "master" source file for the panel image.
- **Copies** The number of copies of the image in the panel.
- **Anchor** The location of the anchor point for the image.
- **Location** The X:Y location of the anchor.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.

Draw

- **Dcode** The D-code used to create the draw.
- **Shape** The shape of the D-code used.
- **Size** The size of the D-code used.
- **From** The X:Y location where the draw begins.
- **To** The X:Y location where the draw ends.
- **Length** The length of the draw.
- **Area** The total square area of the draw.
- **Layer** The layer the item is on.
- **Layer Type** The type of layer the item is on.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.
- **Polarity** The polarity of the item, either positive or negative.
- **Net** The net number that the item belongs to (if any).

Drill

- **Tool** The tool number used for the drill hit.
- **Tool Size** The size of the tool used for the drill hit.
- **Plated** "Yes" means the hit is plated, "No" means it is unplated.
- **Location** The X:Y location of the drill hit.
- **Area** The total square area of the drill hit.
- **Layer** The layer the item is on.
- **Layer Type** The type of layer the item is on.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.
- **NC Group** The NC Group number that the drill hit belongs to.
- **Export Order** The drill hit's order number in the drill sequence. If a value of "-1" is shown, then the order is not set (NC optimization has not been run).
- **Net** The net number that the item belongs to (if any).
- **Operator Message** Indicates whether an operator message is displayed to the machine operator before or after the hit is made.
- **Optional Stop** Indicates if the machine is stopped before or after the hit is made.

Drilled Circle

- **Tool** The tool number used for the circle.
- **Tool Size** The size of the tool used for the circle.
- **Plated** "Yes" means the circle is plated, "No" means it is unplated.
- **Center** The X:Y location of the circle's center point.
- **Diameter** The diameter of the circle.
- **Area** The total square area of the circle.
- **Layer** The layer the circle is on.
- **Layer Type** The type of layer the item is on.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.
- **NC Group** The NC Group number that the circle belongs to.
- **Export Order** The circle's order number in the drill sequence. If a value of "-1" is shown, then the order is not set (NC optimization has not been run).
- **Net** The net number that the item belongs to (if any).
- **Operator Message** Indicates if an operator message is displayed to the machine operator before or after the circle is made.
- **Optional Stop** Indicates if the machine is stopped before or after the circle is made.

Drilled Text

- **Tool** The tool number for the text being queried.
- **Tool Size** The size of the tool used for the text.
- **Plated** "Yes" means the text is plated, "No" means it is unplated.
- **Location** The X:Y location of the insertion point of the text (usually the lower-left corner of the first letter).
- **Text** The letters/numbers used in the text itself.
- **Orientation** The orientation of the line of text: Horizontal or Vertical.
- **Area** The total square area of the text.
- **Layer** The layer the text is on.
- **Layer Type** The type of layer the text is on.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.
- **NC Group** The NC Group number that the text belongs to.
- **Export Order** The text's order number in the drill sequence. If a value of "-1" is shown, then the order is not set (NC optimization has not been run).
- **Net** The net number that the item belongs to (if any).
- **Operator Message** Indicates if an operator message is displayed to the machine operator before or after the text is made.
- **Optional Stop** Indicates if the machine is stopped before or after the text is made.

Flash

- **Dcode** The D-code used to create the flash.
- **Shape** The shape of the D-code used.
- **Size** The size of the D-code used.
- **Location** The location of the item, in X:Y coordinates.
- **Area** The total square area of the flash.
- **Layer** The layer the item is on.
- **Layer Type** The type of layer the item is on.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.
- **Polarity** The polarity of the item, either positive or negative.
- **Net** The net number that the item belongs to (if any).
- **Embedded Passive** If the item is part of an embedded passive, this shows whether it is a termination bar or resistor. If it is a resistor, the reference designator is given.
- **Ohms** If you query an embedded passive resistor, this is the resistance value.

Mill Circle

- **Tool** The tool number for the circle.
- **Tool Size** The size of the tool used for the circle.
- **Plated** "Yes" means the circle is plated, "No" means it is unplated.
- **Center** The X:Y location of the circle's center point.
- **Diameter** The diameter of the circle.
- **Area** The total square area of the circle.
- **Layer** The layer the circle is on.
- **Layer Type** The type of layer the circle is on.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.
- **Feed Rate** The feed rate of the mill tool.
- **Side** Whether the circle is cut on the Inside or Outside of the radius.
- **Direction** Whether the circle is milled in a Clockwise or Counter-Clockwise direction.
- **Length** The distance the mill tool has to travel to create the circle.
- **NC Group** The NC Group number that the circle belongs to.
- **Export Order** The circle's order number in the mill sequence. If a value of "-1" is shown, then the order is not set (NC optimization has not been run).
- **Operator Message** Indicates if an operator message is displayed to the machine operator before or after the circle is made.
- **Optional Stop** Indicates if the machine is stopped before or after the circle is made.

Mill Path

- **Tool** The number of the mill tool used.
- **Tool Size** The size of the tool used.
- **Plated** "Yes" means the path is plated, "No" means it is unplated.
- **Location** The X:Y location where the path begins.
- **Area** The total square area of the draw.
- **Layer** The layer the item queried is on.
- **Layer Type** The type of layer the item is on.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.
- **Compensation** The side of the mill path (as determined by its direction) that compensation was applied to.
- **Feed Rate** The feed rate of the mill tool.
- **Overshoot** The amount added to the last segment of the mill path.
- **Segments** The number of segments in the mill path.
- **Length** The exact distance a mill tool would have to cut to create the path. Any spaces in the mill path created by break tabs are not included in this; however any "legs" from crown breaks are factored in.
- **NC Group** The NC Group number that the path belongs to.

- **Export Order** The path's order number in the mill sequence. If a value of "-1" is shown, then the order is not set (NC optimization has not been run).
- **Operator Message** Indicates if an operator message is displayed to the machine operator before or after the path is made.
- **Optional Stop** Indicates if the machine is stopped before or after the path is made.

Part

- **Ref Des** The selected part's reference designator.
- **Pin 1 Loc** The X:Y location of pin 1.
- **Side** The side of the board the part is placed on.
- **# of Pins** The number of pins in the part.
- **Angle** The part footprint's angle of rotation, if any.
- **Device** The device name.
- **Number** The part number (if any).
- **Value** The value of the device, such as resistance or capacitance.
- **Tolerance** The precision of the device's value, usually a percentage.
- **Description** Information (if any) about what the device is.
- **Ref Des Loc** The X:Y location of the part's reference designator.
- **Footprint** The part footprint's name.
- **Centroid Loc** The X:Y location of the part's centroid.
- **Insertion Pt** The X:Y location of the part's insertion point.

Polygon

Only raster polygons are identified as "Polygon". Vector polygons are identified by their individual draws.

- **From** The X:Y location where the polygon border begins.
- **To** The X:Y location where the polygon border ends (always the same as the beginning).
- **Closest Pt** The closest vertex point to the coordinate selected when you queried the polygon.
- **Num Pts** The number of points on the polygon border.
- **Area** The square area of the entire polygon.
- **Layer** The layer the polygon is on.
- **Layer Type** The type of layer the polygon is on.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.
- **Polarity** The polarity of the polygon, either positive (Dark) or negative (Clear).
- **Net** The net number that the item belongs to (if any).

Polyline

Polylines are contiguous lines made of multiple draws.

- **Dcode** The D-code used to create the polyline.
- **Shape** The shape of the D-code used.
- **Size** The size of the D-code used.
- **From** The X:Y location where the polyline begins
- **To** The X:Y location where the polyline ends.
- **Closest Pt** The closest start/end point of the draw within the polyline to the coordinate that you queried.
- **Num Pts** The number of draw start/end points within the polyline.
- **Length** The length of the polyline.
- **Area** The total square area of the polyline.
- **Layer** The layer the polyline is on.
- **Layer Type** The type of layer the polyline is on.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.
- **Polarity** The polarity of the polyline, either positive (Dark) or negative (Clear).
- **Net** The net number that the item belongs to (if any).

Slot

- **Tool** The tool number for the slot.
- **Tool Size** The size of the tool used for the slot.

- **Plated** "Yes" means the slot is plated, "No" means it is unplated.
- **From** The X:Y location where the slot begins.
- **To** The X:Y location where the slot ends.
- **Length** The length of the slot.
- **Area** The total square area of the slot.
- **Layer** The layer the item is on.
- **Layer Type** The type of layer the item is on.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.
- **NC Group** The NC Group number that the slot belongs to.
- **Export Order** The slot's order number in the drill sequence. If a value of "-1" is returned, then the order has not been set (NC optimization has not been run).
- **Net** The net number that the item belongs to (if any).
- **Operator Message** Indicates if an operator message is displayed to the machine operator before or after the slot is made.
- **Optional Stop** Indicates if the machine is stopped before or after the slot is made.

Text

If the text is created with a font other than "GerbTool-Stroke", the font is noted in the Type field.

- **Dcode** The D-code used to create the text.
- **Shape** The shape of the D-code used.
- **Size** The size of the D-code used.
- **Location** The X:Y location where the text begins.
- **Area** The total square area of the text.
- **Layer** The layer the item queried is on.
- **Layer Type** The type of layer the item is on.
- **SeqNo** The item's order in the Gerber database. Item 1 is the first item listed in the Gerber file, item 2 is the second, etc.
- **Polarity** The polarity of the text, either positive (Dark) or negative (Clear).
- **Net** The net number that the item belongs to (if any).

Status Bar

The Status Bar, at the bottom of the desktop, provides specific command instructions and feedback. During certain commands, such as those where you are converting one type of data item to another, an Enable/Disable Prompts button appears on the left side. Disabling Prompts will prevent confirmation message boxes from appearing, after you select an item -- the data are converted immediately after selection.

Other useful information is also displayed on the right side of the status bar, in the following order:

Database Modified Indicator

"MOD" appears if you have made any modifications to the currently loaded database.

Current Folder

The directory path of the currently loaded database is displayed.

Redraw Status

Automatic redraws of the workspace can be turned off, using the Options|Configure command. When you select the Minimize Redraws option, you control when the workspace is refreshed, by pressing the R hotkey.

Undo Status

The Undo command allows you to reverse edits that you have just made. As useful as this is, the Undo memory buffer can consume memory and affect system performance. You can disable the Undo function using the Options|Configure command.

DRC Status

The number of existing Analysis errors is shown.

Select Group

The number of items in the current Select Group (Sel) is shown.

Units of Measure
The current units of measure are displayed.

Process Indicator
The process indicator light, in the lower-right corner of the desktop, tells you if the system is currently processing data. If the light is flashing red, VisualCAM is currently processing data. When the light is solid green, VisualCAM is not processing data. If the light is solid red (not flashing), a system error has occurred and VisualCAM is unable to process data.

Workspace
The Workspace is where the graphical representation of your database appears. The commands in the Menu Bar (and their associated toolbar buttons) control your view of the workspace and allow you to edit the data. You can change the background color of the workspace using the Options|Configure command.

Anytime VisualCAM is redrawing the display or highlighting a window of data, you may halt the drawing process by pressing the Esc key. This will not effect the operation of a command, and in many cases will speed up the operation of a command. If you would like to terminate a redraw using the right mouse button, you must disable the right-click shortcut menu. See the Options|Configure command.

Grid
A grid can be displayed in the workspace, by selecting the View|Grid command. The size of the grid, as well as the units of measure used for your database, can also be controlled using the Options|Configure command.

View Tabs
At the bottom of the Workspace are View Tabs. The default tabs are Log and Main. Log shows a record of all your actions since you opened or created a database, and is cleared when you select the File|New or File|Open command. To save the log as a file, right-click in the Log screen and select Save Log from the shortcut menu, or select the File|Save Log command. Main shows the graphical display of your database.

You can add additional tabs, maintaining one view of your database in each, so that you are not required to frequently zoom and pan your view. You can perform edits, such as copy data, from one view to another. You can also delete tabs which you have added.

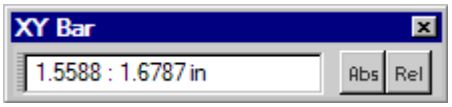
Cursor
The cursor, or "cross hair", is used to select objects, commands, buttons, etc. in the workspace. Moving your mouse moves the cursor accordingly. There are several ways that you can modify the behavior of the cursor, using hotkeys and the Options|Configure command.

Each mouse button has a unique function, which can be customized. By default, coordinates showing the current location of the cursor are attached to it. You can turn off this display, as well as change the size of the cursor.

Film Box
The film box shows the extents of your film. The film size, as well as its display color, are stored in your saved .vcam files, and can be changed using the Options|Configure command.

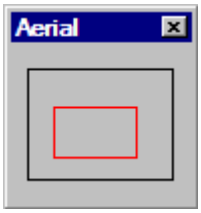
The View|Film Box command allows you to zoom so that the extents of the film box are displayed in the workspace.

XY Bar
This toolbar shows the X and Y coordinates of your cursor in the workspace.



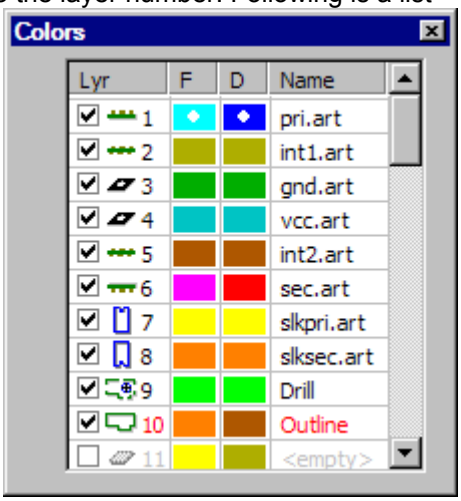
The Abs and Rel buttons allow you to change the coordinates to Absolute or Relative. These buttons are available when you are in one of the Edit or Add functions.

Aerial Bar
The Aerial view allows you to tell, at a glance, exactly where your current view window is located in reference to your data. The red rectangle represents your current view (what appears in the workspace), and the black rectangle represents the database extents. You can close the Aerial bar by clicking on the X in the upper-right corner. You can expand the size of the bar to the entire height of the workspace by clicking the up-arrow; then reduce its size by clicking the down-arrow. While it is docked in the VisualCAM window, you can resize the height and width of the bar by moving your cursor to one of its outer edges. When the cursor changes to ↔, click and hold the left mouse button while you drag the edge to the desired size.



Color Bar

The Color Bar is available at all times to change active layer, layer colors and visibility. Layer names are also presented for your reference, and layer types are represented by the icon next to the layer number. Following is a list of icons and the layer type they represent.



- Top
- Inner
- Bottom
- Plane
- Silk Bottom
- Silk Top
- Mask Top
- Mask Bottom
- Paste Top
- Paste Bottom
- Border
- NC (Drill/Mill)
- Composite
- Drawing
- Insulator
- Etch (Subtractive) Passive
- Screen (Additive) Passive
- External Netlist (can only be set by VisualCAM after importing a netlist)
- Other

If a layer contains item-level polarity, the Draw and Flash colors contain a diamond shape, as shown for layer 1 in the above illustration.

If you wish to change a layer name or type, use the Layer table or Navigator. You can close the Color Bar by clicking on the X in the upper-right corner. You can expand the size of the bar to the entire height of the workspace by clicking the up-arrow; then reduce its size by clicking the down-arrow. While it is docked in the VisualCAM window, you can resize the height and width of the bar by moving your cursor to one of its outer edges. When the cursor changes to ⇹, click and hold the left mouse button while you drag the edge to the desired size.

Visibility

Visibility for individual layers, layer sets and composites is controlled by clicking in the check box next to the desired layer number. A check mark indicates visibility, in which state a layer can be edited; no check mark indicates the layer is not visible, and cannot be edited.

To turn all layers on or off, right-click in the Color Bar, and select either All On or All Off. When you select All Off, all layers except the active layer are turned off.

Setting the Active layer

Double-clicking on a layer number makes the layer "active". You can also right-click on the desired layer in the Color Bar, and select Set Active from the shortcut menu. The active layer appears on top of the other layers in the workspace (this does not change the actual layer order).

The active layer can also be set using the Layer Bar.

Color Selector



You can control the color of flashes and draws for each of your layers. To change one of the colors, click on it. The Color Selector appears, and you can select your desired color from the available selection. To change the color choice at a particular location in the Color Selector, right-click on the desired color. This opens the standard Windows color selector that allows you to define a color.

To dismiss the Color Selector without choosing a color, press the Esc key.

If you have a specific set of layer colors you would like to use for all your designs, you are provided with a file in the Macros folder called demo.mac, which includes a macro called SetLayerColors. Use the following steps to utilize this macro.

1. Open the demo.mac file the Macro Developer, and edit it to specify your desired colors. Save the file and return to the main VisualCAM window.
2. Select the Macro|Load command and load the demo.mac file.
3. Select the Macro|Run command, and run the SetLayerColors macro.

Tip: If you wish to run the SetLayerColors macro automatically when VisualCAM starts, rename it to OnStartup. To run it whenever you select the File|Open command, rename it to OnOpen.

Hotkeys

Hotkeys (also known as "nested commands") are available anytime VisualCAM has prompted you to enter a point or is idle. All hotkeys are executed immediately without affecting the current command. Some hotkeys are a toggle, which means that pressing the key will turn a mode on or off.

When a hotkey has an equivalent menu command, it is indicated. Equivalents for most hotkeys are also available through a right-click, context sensitive, shortcut menu, if enabled.

Key	Command	Key	Command
1-9,0	Bring a layer (1-10) to top	**O or -**	View|Zoom Out
Ctrl+1-9,0	Bring a layer (11-20) to top	**Ctrl+O**	File|Open
A	Turn on all layers	**P**	View|Pan
Ctrl+A	Turn on only active layer	**Ctrl+P**	Toggle Autopan mode (pan with cursor)
Shift+A	Setup|Apertures	**Ctrl+Alt+Q**	Exit without confirmation
B	Toggle View|Backside	**Q**	Query|Item
Shift+B	Setup|Layer Sets|Blind/Buried	**R**	View|Redraw
C	Enter absolute coordinates	**Ctrl+R**	View|All
Ctrl+C	Enter relative coordinates	**S**	Toggle Options|Grid Snap
D	Increment current D-code/tool	**Ctrl+S**	File|Save
Ctrl+D	Decrement current D-code/tool	**T**	Toggle tool path view
F	View|Selection Filter	**Shift+T**	Setup|NC Tools
Ctrl+F	Edit configuration flags	**U**	Undo last edit
G	Toggle View|Grid	**Ctrl+U**	Undo all edits
Ctrl+G	Edit grid settings	**Shift+U**	Reverse last undo (Redo)
H	Toggle View|Highlights	**V**	Toggle View|Composites
I or +	View|Zoom In	**Ctrl+V**	Toggle View|Virtual Panel
Ctrl+I	Copy workspace to Clipboard	**Y**	Setup|Layers
L	Increment active layer	**Esc**	End current function
Ctrl+L	Decrement active layer	**Enter**	Enter coordinate at cursor location
Shift+L	Setup|Layer Sets|View/Edit	**Home**	Snap cursor to item center
M	Macro|Run	**Ctrl+Home**	Snap cursor to item center & show Absolute Coordinates box
Ctrl+M	Options|Units/Precision	**PgUp**	Increase cursor bounding box
N	Toggle active layer display positive/negative	**PgDn**	Decrease cursor bounding box
Ctrl+N	File|New	←→↑↓	Scroll left, right, up, and down

VisualCAM also offers competitive product hotkey support. If you wish to apply VisualCAM functions to the equivalent hotkey used in one of the competitive products listed, you can change the setting using the Options|Configure command. Only the competitive product hotkeys listed below are supported.

VisualCAM Function [Hotkey]	CAM350 Hotkey	ViewMaster Hotkey
Bring layer 1-10 to the top [**1-9,0**]	**1-9,0**	*None*
Setup|Apertures [**Shift+A**]	**A**	**F5**
Snap cursor to center of item [**Home**]	*None*	**A**
Zoom in at current cursor position	**C**	*None*
Set active D-code, by number	**D**	**Shift+D**
Query|Measure|Point To Point	*None*	**D**
View|Sketch	**F**	**Shift+O**
Toggle workspace redraw on/off	**G**	*None*
View|Grid [**G**]	**V**	**G**
View|Highlights [**H**]	**H**	*None*
Invert current select group	*None*	**Ctrl+I**
Edit|Copy	*None*	**K**

VisualCAM Function [Hotkey]	CAM350 Hotkey	ViewMaster Hotkey
Turn layers off, by number (0 = all)	**K**	*None*
Turn layers on, by number (0 = all)	**L**	*None*
Edit|Move	*None*	**Shift+M**
Toggle active layer display positive/negative [**N**]	**N**	*None*
File|New	*None*	**Ctrl+N**
Toggle orthogonal snap (0/45/90)	**O**	*None*
File|Open	*None*	**Ctrl+O**
Zoom to previous position	**P**	*None*
File|Print	*None*	**Ctrl+P**
Query|Item [**Q**]	**Q**	**Shift+A**
Edit|Align Layers	*None*	**Ctrl+Q**
View|Redraw [**R**]	**R**	**End**
Edit|Rotate	*None*	**Ctrl+R**
Options|Grid Snap [**S**]	**S**	*None*
Edit|Select|New Group	*None*	**S**
File|Save	*None*	**Ctrl+S**
View|Overlay	**T**	*None*
Transcode current item	*None*	**Ctrl+T**
Edit|Undo [**U**]	**U**	**Ctrl+Z**
Edit|Redo [**Shift+U**]	**Ctrl+U**	*None*
Edit|Select|Clear	*None*	**Ctrl+U**
Zoom by Window (default left mouse button*)	**W**	**Shift+Home**
Edit|Select|New Group (Window mode)	*None*	**Ctrl+W**
Edit|Mirror (Group mode)	*None*	**Shift+X** / **Shift+Y**
Setup|Layers [**Y**]	**Y**	*None***
Toggle all layers on [**A**]	*None*	**0 / Ctrl+0**
Toggle all layers off, except active [**Ctrl+A**]	*None*	**Shift+0**
Help [**F1**]	**F1**	**F1**
File|Import|Gerber	*None*	**F2**
File|Import|Import Wizard	*None*	**Ctrl+F2**
Exit program without prompt [**Ctrl+Alt+Q**]	*None*	**Alt+F4**
Setup|NC Tools [**Shift+T**]	*None*	**Shift+F5**
View|Zoom In [**+**]	**+**	**PgUp**
View|Zoom Out [**-**]	**-**	**PgDn**
Make next layer active/visible [**L**]	None	**=**
Make previous layer active [**Ctrl+L**]	*None*	**Backspace**
View extents of all on layers [**Ctrl+R**]	**Home**	**Home**
Pan to cursor location [**P**]	**Ins**	*None*
Edit|Delete	*None*	**Del**
Increase snap box size [**PgUp**]	**PgUp**	*None*
Decrease snap box size [**PgDn**]	**PgDn**	*None*
End current command [**Esc**]	**Right mouse button*/Esc**	**Esc**
Select item	**Spacebar**	**Enter**
Pan left [←]	*None*	**Ctrl+←**
Pan right [→]	*None*	**Ctrl+→**
Pan up [↑]	*None*	**Ctrl+↑**
Pan down [↓]	*None*	**Ctrl+↓**

*To change the default mouse functions, use the Function Key/Mouse Button tab in the Options|Configure command.

**Because the F10 key is reserved by a Windows function, the F10 hotkey from ViewMaster, which opens the Layer Table, cannot be emulated.

Related topics

Mouse and Function key commands

Mouse and Function Key Commands

Mouse Functions

The term "click" refers to the selection of an item, command, or control by placing the mouse cursor over it, then pressing and releasing the left mouse button. If you do not have a command function active, you can use the left mouse button to create a View Window and zoom in on an area of your workspace.

To "double-click" means to perform the above action, but pressing and releasing the left mouse button twice in rapid succession.

The term "right-click" refers to the selection of an item, command, or control by placing the mouse cursor over it, then pressing and releasing the right mouse button. Often, as when you right-click in the main workspace, a shortcut menu appears, allowing you to select a command by clicking on it.

Clicking the middle mouse button in the main VisualCAM workspace is the equivalent of the View|Zoom In command.

You can use the Options|Configure command to change the default left, right, and middle mouse button assignments described above.

Function Keys

VisualCAM comes pre-configured with the following function key assignments:

Key	Assignment
F1	Help
F2	View|Film Box
F3	View|Previous
F4	View|All
F5	Setup|Layers
F6	Setup|Apertures
F7	Documentation|Reports|Apertures
F8	Query|Highlight
F9	Query|Item
F10	Menu
F11	Edit|Select|Add To
F12	Edit|Select|Remove From

You can change any of the default function key assignments listed above, using the Options|Configure command.

See *Hotkeys* for a list of keyboard shortcut keys.

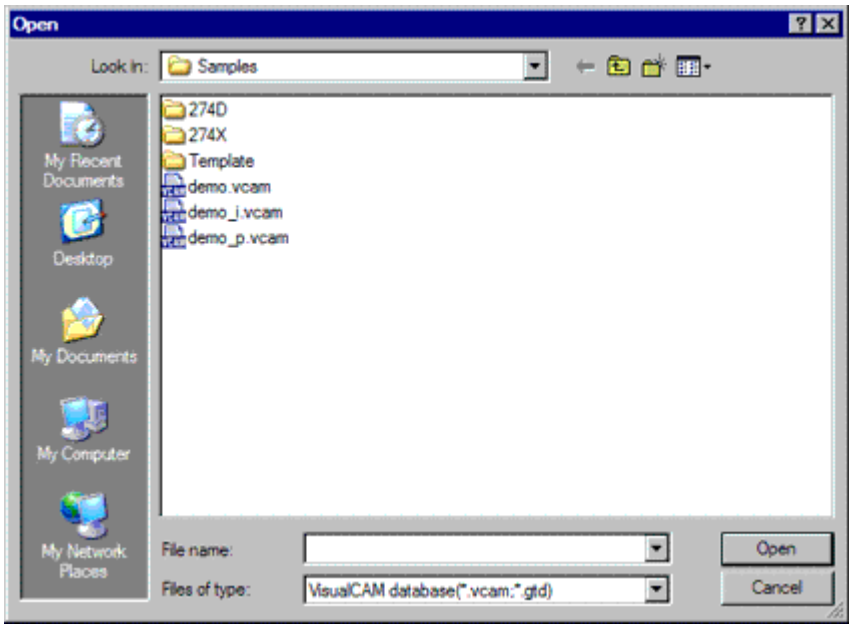
Dialog Boxes

Dialog boxes are used to enter information into VisualCAM. They may contain data entry fields, selectable options, color buttons, help buttons, scroll bars and exit buttons (OK, Cancel, and Close).

Most VisualCAM dialog boxes have a help button with a question mark located in the title bar. To use this feature, click the help button and then click on a control within the dialog box. A popup window will appear with information specific to that control and dialog box.

File Selection

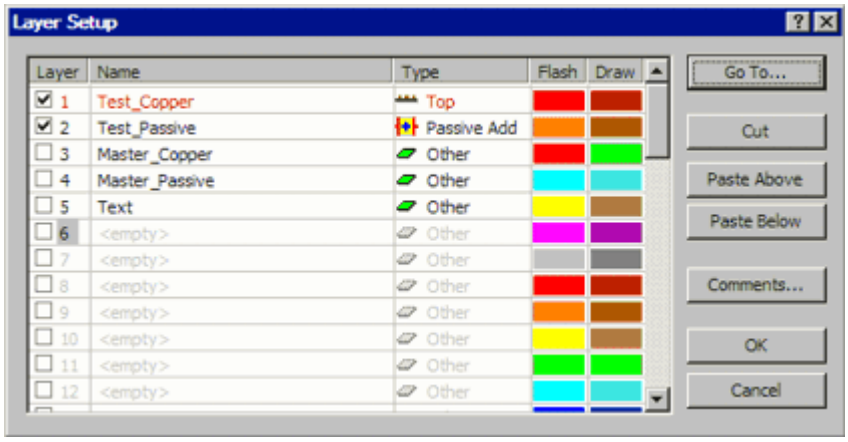
The File Selection dialog box provides a convenient way of locating files so you do not have to remember them all. There are two forms of the File Selection dialog box. The first (below) allows you to select only one filename. You select the desired file by clicking directly on a filename. The chosen filename appears in the File Name field at the bottom of the dialog box.



The second allows you to select multiple filenames by holding down the Ctrl key on your keyboard, and clicking on each desired filename. Each selection remains highlighted, and appears in the File Name field. An example of this type of dialog box is the one used for File|Import|Barco DPF.

Edit Forms

Edit forms are used to enter information into VisualCAM. They may contain data entry fields, checkable buttons, color buttons, scroll bars and exit buttons (*OK, Cancel*, etc.). This type of form is exemplified by the Setup|Layers dialog box, as shown below.



Accessing Help

In addition to this Help system, there are many ways to obtain more information about using VisualCAM.

Most dialog boxes offer context sensitive help, which can be accessed two ways.

- Right-click on any dialog box control. If help is available for it, "What's This" appears next to your cursor. Click on What's This to view information about the control.
- Click on the "?" button in the upper-right corner of the dialog box, then on the dialog box control in question. Explanatory text appears.

The VisualCAM web site, http://www.wssi.com/visualcam, offers a support page with documentation in PDF format, FAQ's with video demonstrations, and other helpful information. You can also join the VisualCAM User Forum, where you can ask questions and discuss VisualCAM with other users.

Performance Tips

Hotkeys Execute Immediately

Nested commands, otherwise known as "hotkeys", are available at all times when VisualCAM is waiting for you to enter a coordinate (point) or is idle (i.e. no command has been selected). With these commands, you can move around, snap to the center of a database item, change what layers are viewed, undo edits, etc.

Interrupting Redraws and Highlights

Any command that redraws the database or highlights a group of items can be sped up by canceling the drawing process. By pressing the Esc key, you can halt whatever is redrawing the display. This does not effect the operation of the command; only the redraw is effected. Once you are comfortable with the operation of various functions, you will find that this ability significantly speeds things up.

Tip: You can have the right mouse button end functions like the Esc key by disabling the Right Click Menu.

Undoing Edits

The Undo command provides a high level of freedom when making database edits. When Undo is enabled, you may experiment with edits without fear of data loss. Since undo is available as the U hotkey you may undo edits immediately without even having to exit the current command. Undo works for all edits regardless of size, and there is no limit to the number of edits you can undo. Just remember to enable Undo with the Options|Configure/General command *before* making your edits. Then use the Edit|Undo command, the toolbar button, or the U hotkey to undo changes as desired.

Using the Selection Filter

The Selection Filter is available during most Edit functions. If you are working with a dense design, the selection filter helps you filter out the items that you do not want to select. It also allows you to select multiple items and items on specific layers.

When you are not in a command that uses the Selection Filter, the text in the Selection Filter is gray and you cannot select any options. You also do not have the option of using the View|Selection Filter command or the F hotkey when not in a function that uses the Selection Filter. If you undock the Selection Filter from the VisualCAM window while not in a function that uses it, it will disappear. It will automatically appear again when a function requires it.

You can close the display at any time by clicking on the X in the upper-right corner.

Copying to the Windows Clipboard

The Copy Selection To Clipboard command it is very effective for copying sections of a board (as many layers as needed) and pasting to a relative distance from the copy location within the same design. When used with the Selection Filter, it has the benefit of copying multiple items from multiple layers, and copying them to relative locations on the same layer numbers in the same design *or* a new design. Edit|Copy puts *every* copied item on *each* destination layer, so copying from several layers at a time puts everything onto one layer.

Programming Your Mouse and Function Keys

VisualCAM's easy-to-use Graphical User Interface (GUI) is further enhanced with the versatility of programmable mouse and function keys. Using the Options|Configure dialog box, you may program both the mouse buttons and function keys F1-F12 with commands that you frequently use and in a layout that you find comfortable.

Memory Considerations

VisualCAM was developed to operate in true 32-bit protected mode with virtual memory. This allows VisualCAM to address the entire memory range of the CPU, even if the actual installed amount of memory (real memory) is less.

While virtual memory is a very powerful feature, there is no substitute for real memory for maximum speed. You should also keep in mind that other applications that you may have running on your computer, such as e-mail programs and other utilities that run in the background, often use a significant amount of memory. This can cause performance problems in VisualCAM, because not enough memory may be available to process large amounts of data. For example, if you have no other applications running on a 256 MB system, 20 MB of Gerber files will load quickly into VisualCAM. If you load those same files while other applications are running, you may notice a lot of disk activity as the virtual memory manager begins to "thrash" due to the disproportionately small amount of real memory available.

If you receive an error message, such as "memory allocation error," this indicates that your system has exhausted its allocated swap space. You can help keep the swap file usage down by occasionally using the Edit|Purge command or disabling the Undo feature, if it is not required. Purging compacts VisualCAM's internal database and allows more efficient use of memory.

Tutorials

These tutorials help you perform various tasks with VisualCAM. They include step-by-step instructions, and explanations of various concepts.

Using Files

When you save a database using the VisualCAM Save command, all the design data are contained within the single file (any original Gerber files remain untouched and are no longer needed). The File|Open command is used for opening VisualCAM database files.

Creating a New Design

To create a new design, select the File|New command. This clears your workspace and allows you to begin a new (and initially untitled) design. You may begin adding data manually or importing files into your design.

When you are ready to save your new design for the first time, VisualCAM will prompt you to choose a new filename for your design.

Opening/Importing

Any file that is not a VisualCAM (.vcam) or GerbTool (.gtd) database must be imported into VisualCAM, using an appropriate command. The following list shows supported file formats, and what command is used to import them. You can find sample files in many of the below formats in the Samples folder of your VisualCAM program folder.

The Import Wizard will recognize all of the "importable" files listed below. If you are in doubt about a file's format, the Import Wizard will tell you. If you know you are importing a non-Gerber file, such as a DXF file, we recommend you use the import command specific to that format. (It will save you the time of having to step through the various Import Wizard dialog boxes.)

When importing 274-D Gerber files, external aperture list files are required. VisualCAM provides aperture list conversion for most of the popular CAD and photo-plotter aperture list formats in use today. This conversion process will translate a CAD aperture list directly into VisualCAM, reducing data entry related problems.

File Type	Import Command
Aperture List	File|Import|Import Wizard *or* File|Import|Aperture List
Barco	File|Import|Barco DPF
CAM350 v7.X or earlier	File|Import|CAM350 *or* File|Open
Custom Aperture Library	Setup|Apertures /Custom Ap /Load From Lib
DXF (AutoCAD 2000 or earlier)	File|Import|DXF
Gerber (274-D, 274-X, Fire9XXX)	File|Import|Import Wizard *(recommended) or* File|Import|Gerber
GerbTool (.gtd)	File|Open
HPGL	File|Import|HPGL
IPC-2581 (Offspring)	File|Import|IPC-2581 (Offspring)
IPC-D-356/A	File|Import|Netlist|IPC-D-356
NC (Drill/Mill)	File|Import|Import Wizard *or* File|Import|NC (Drill/Mill)
NC (Drill/Mill) Tool List	File|Import|NC Tool List
ODB++	File|Import|ODB++
ODB++ Netlist	File|Import|Netlist|ODB++
ODB++(X)	File|Import|ODB++(X)
ODB++(X) Netlist	File|Import|Netlist|ODB++(X)
PADS PowerPCB ASCII File	File|Import|PADS ASCII
PADS PowerPCB ASCII Netlist	File|Import|Netlist|PADS ASCII
VisualCAM (.vcam)	File|Open
VisualCAM Analysis Rule (.rul)	Analysis|DRC/MRC /Load
VisualCAM Character Set Definition (.csd)	Tools|Convert|Drawn Text /Load
VisualCAM Footprint Library (.fpl)	Assembly|Setup Footprint Library /Load *or* Setup|Footprint Library /Load
VisualCAM NC Tool Table (.nct)	Setup|NC Tools /Load
VisualCAM Report (.txt)	*Use any text editor (Notepad, WordPad, etc.)*

There are also files that are used by VisualCAM, but do not need to be imported or opened; you simply need to tell VisualCAM where they are located, in case they need to be used. If any of the below files are provided for you in the VisualCAM program folder, the name of the sub-directory is indicated:

File Type	Command

Aperture List Automatic Conversion Rule (ACR)	Options|Configure /Ap List Converters. *Several .acr files are provided for you in the ApConv folder.*
Bill of Materials (BOM) Automatic Conversion Rule (ACR)	*ACR files for importing BOMs, using* Assembly|Import BOM, *are placed in the BOMConv folder.*
Centroid File Automatic Conversion Rule (ACR)	*ACR files for importing centroid information, using* Assembly|Parts|Identify Using|Centroid File, *are placed in the CentroidConv folder.*
DXF Fonts	File|Import|DXF. *Several .shx files are provided for you in the DXFFonts folder.*
Macros	Macro|Load (*execute using* Macro|Run). *Several Macro (.mac) files are provided for you in the Macros folder.*
NC Tool List Automatic Conversion Rule (ACR)	*Any file placed in the ToolConv folder is automatically detected by* File|Import|NC Tool List.
Text Fonts	*The font file used by* Add|Text *and other text commands is in the Fonts folder.*

Drag-and-Drop 274-X and .vcam Files

All the necessary aperture information is contained within 274-X files, and VisualCAM database (.vcam) files contain all the necessary information for an entire design. Because of this, it is a simple task for VisualCAM to load these files. VisualCAM offers a simple "drag-and-drop" feature for importing 274-X files and opening VisualCAM files, allowing you to view your data quickly.

1. Open VisualCAM and the Windows File Explorer.
2. In Windows Explorer, find the 274-X or .vcam file you wish to import.
3. Click on the file and hold down your mouse button. Drag the cursor to the VisualCAM workspace, and release the mouse button. The data immediately appears in the workspace.

Troubleshooting Gerber Import

The Import Wizard automatically detects the format of your Gerber files, and imports them accordingly. Some issues with importing Gerber data and aperture tables may occur, however, so below are some tips for making sure your data are imported properly.

- Since Gerber files contain only numbers with no decimal point, a number such as X12345 could mean 1.2345 inches, 12.345 inches, or 123.45 inches. If any of your files came in too large or too small, you need to start the import process again and select a more appropriate M.N Format for your files.
- If the M.N setting is correct but a file is still not correctly imported, check to make sure the Zero Suppression setting is correct. Zeros are added to numbers in a file to make sure they fit the required M.N setting. If a number is X01234, and you use 2.3 format but suppress leading zeros, a 1.234 file will import as 12.34.
- If there is an incorrect Coordinate Mode setting, files usually read in as a long jagged line or sunburst. Absolute coordinates are used when the locations refer to a common origin (0,0). Incremental coordinates are when each coordinate is a displacement from the previous. Double-check this setting for any errors.
- Your data may be imported correctly, but the layers are not aligned properly or the origin is set far away from the actual data extents. This is often a problem in Gerber files exported from PADS PowerPCB, which (by default) centers layers based on a page (film) format size, with the origin is based on the lower-left corner of the page. You can fix this by disabling the default setting in PowerPCB. Otherwise use the Edit|Align Layers command to align the layers, and Edit|Origin to change the origin.

Saving/Exporting

The File|Save and File|Save As commands are used to save VisualCAM database (.vcam) files. To save to any file format that is not a VisualCAM database (such as Gerber), you must use an appropriate Export or Save command. The following list shows supported file formats, and what command is used to export them.

File Type	Export Command
Aperture List	File|Export|Aperture List
Aperture Report	Documentation|Reports|Apertures /Save
Barco	File|Export|Barco DPF
Bill of Materials	Documentation|Reports|BOM /Save *or* Assembly|Parts|BOM Report /Save
Bitmap	File|Export|Bitmap
Custom Aperture	Custom Aperture Editor File|Save
Custom Aperture Library	Setup|Apertures /Custom Ap /Save To Lib

File Type	Export Command
DXF (AutoCAD 2000)	File|Export|DXF
FATF	Assembly|Export FATF
Gerber (274-D, 274-X, Fire9XXX)	File|Export|Gerber
GerbTool 15.X (.gtd)	File|Export GerbTool
HPGL	File|Export|HPGL
IPC-2581 (Offspring)	File|Export|IPC-2581 (Offspring)
IPC-D-350	File|Export|IPC-D-350
IPC-D-356/A	File|Export|IPC-D-356
Macro	Macro Developer File|Save
NC (Drill/Mill)	File|Export|NC (Drill/Mill)
NC (Drill/Mill) Tool List	File|Export|NC Tool List
NC (Drill/Mill) Tool Report	Documentaton|Reports|NC Tools /Save
Netlist	Tools|Netlist|Save
ODB++	File|Export|ODB++
ODB++(X)	File|Export|ODB++(X)
PostScript	File|Export|PostScript
Test Fixture, Bed of Nails	Tools|Test Points|Test Fixture|Save Files
Test Fixture, Flying Probe	Tools|Test Points|Flying Probe|IPC-D-356A
VisualCAM (.vcam)	File|Save or File|Save As
VisualCAM Character Set Definition (.csd)	Tools|Convert|Drawn Text /Save
VisualCAM Footprint Library (.fpl)	Assembly|Setup Footprint Library /Save *or* Setup|Footprint Library /Save
VisualCAM NC Tool Table (.nct)	Setup|NC Tools /Save
VisualCAM Analysis Rule (.rul)	Analysis|DRC/MRC /Save

Related topics

Gerber for Beginners
ACR Files
Aperture List Files
Sample VisualCAM Netlist (.nl) File
Text Font Files
Color List File

Gerber for Beginners

While it is not necessary to understand the syntax of a Gerber file in order to import and manipulate your data in VisualCAM, a basic understanding of Gerber is helpful in understanding the elements of your design. This tutorial is intended to be a guide to understanding the fundamentals of Gerber, and how it relates to your use of VisualCAM. If you would like a more thorough explanation of the Gerber 274-X format, we recommend the "Gerber RS-274X Format User's Guide" by Barco Graphics N.V. That document was used as a primary information source for this tutorial.

When you import Gerber or any other data file into VisualCAM, VisualCAM reads the information and stores it in its own internal database format. When you manipulate the data, you are not manipulating your original files. You can save your data in a VisualCAM .vcam file, which is a single file that contains all your data, or you can export your files as Gerber or any of the supported file formats listed in the File|Export menu.

Apertures and D-codes

Two types of Gerber data (274-D, and 274-X) can be used to create individual, photoplotted layers of PCB artwork. The original Gerber formats were designed for use by vector photoplotters. These photoplotters selected individual aperture shapes from an aperture wheel, exposed the aperture to film by opening and closing a shutter. The Gerber file provided the instructions to the photoplotter, telling it which shape to place and where to place it. When an aperture was "flashed", the shutter merely opened and closed in a single location on the film. When an aperture was "drawn", the shutter opened, the aperture was moved across the film, and then the shutter closed, thus creating a line segment. Thus a Gerber file is essentially an ASCII file with a series of commands instructing the photoplotter where to move, when to open and close the shutter, and which aperture to use.

In order for this to occur, the photoplotter needs an aperture list to define the shapes, or D-codes, used. The aperture list cross-references the aperture number with a D-code of the desired shape and size. Each aperture is assigned a number (which represented its position on the wheel). In the 274-D format, the aperture list is a separate file. Following is an example of an aperture list:

```
Aper   Dcode   Type      Size-X   Size-Y
1      10      Round     10       0
2      11      Round     5        0
3      12      Round     20       0
4      13      Round     12       0
5      14      Round     75       0
6      15      Round     50       0
7      16      Round     80       0
8      17      Round     100      0
9      18      Round     60       0
10     19      Square    60       60
11     72      Target    200      0
12     73      Thermal   80       60
```

Aperture lists do not always appear as the above example. Each CAD/CAM system can use whichever format they desire. Because of this, when VisualCAM imports an aperture list, an Automatic Conversion Rule (ACR) file is necessary to interpret the aperture list. VisualCAM provides a GUI-driven ACR File creator that allows you interactively create an ACR file for any aperture list, rather than creating it manually in a text editor.

There are a few things you will notice in the aperture list. Aperture numbers are always listed sequentially, although the D-codes are not. You will also notice that the D-code numbers begin with "10". This is because D01, D02, and D03 are special D-codes used to control the drawing and flashing of the apertures. Only D-codes 10 and higher are used for the aperture shapes.

D-code	Function
D01 or D1	Shutter open (begin draw).
D02 or D2	Shutter closed.
D03 or D3	Flash.
D10 and higher	Apertures.

When using VisualCAM to view your 274-D Gerber files, you must import both the Gerber files and the aperture list files. You can subsequently use the Setup|Apertures command to view and edit your aperture list, if necessary. If you do not import the aperture list files, you will be warned that your apertures are undefined. The VisualCAM aperture table will contain the D-code references from your Gerber file, but you will have to assign the shapes and sizes.

Because 274-D does not have the notion of a "custom aperture", if you export an aperture list from VisualCAM that contains custom apertures, the custom apertures are listed but not completely defined (only their size is indicated). If you wish, prior to exporting, you can "explode" your custom apertures into their individual elements by using the Edit|D-Code|Explode Customs command. This essentially removes all customs from your database. Otherwise, you

can save your custom apertures to separate library files, which contain their complete information. When you re-import a VisualCAM aperture list that contains custom apertures, they are defined as "moires". You will then have to use the Custom Ap button in the Aperture Setup dialog box to reload or define your custom apertures. Use of 274-X will avoid these issues, because the custom aperture definitions are contained within the 274-X data.

G-codes

Now that we understand how to identify the apertures that are used in the Gerber file, we have to tell the photoplotter what to do with them. G-codes are used for this purpose.

G-code	Function
G00	Move
G01 or G1	Linear Interpolation (1X scale)
G02 or G2	Clockwise Circular Interpolation
G03 or G3	Counterclockwise Circular Interpolation
G04 or G4	Ignore Data Block (used for comments)
G10	Linear Interpolation (10X scale)
G11	Linear Interpolation (0.1X scale)
G12	Linear Interpolation (0.01X scale)
G36	Turn On Polygon Area Fill
G37	Turn Off Polygon Area Fill
G54	Tool Prepare
G70	Specify Inches
G71	Specify Millimeters
G74	Disable 360° Circular Interpolation (single quadrant)
G75	Enable 360° Circular Interpolation (multi-quadrant)
G90	Specify Absolute Format
G91	Specify Incremental Format

You will not see all of these G-codes in a 274-D file. They simply aren't required. You will normally only see G01 and G54.

Linear interpolation plots a straight line from the present position to the X:Y coordinate specified by the data block. The format of a G01 command is:

G01X<X coordinate>Y<Y coordinate>D<D-code number>*

Circular interpolation is used to create arcs and circles. The G02 and G03 commands control single quadrant (90°) interpolation, which produces an arc. The G75 command controls multi-quadrant (360°) interpolation, which produces a circle. The G74 command disables interpolation.

G<nn>X<X coordinate>Y<Y coordinate>I<X distance>J<Y distance>D<D-code number>*

Parameter	Description
G<nn>	The G-code number (G02, G03, or G75)
X<X coordinate>Y<Y coordinate>	Defines the arc end point. The starting point is the current tool position (wherever the last command left it).
I<X distance>J<Y distance>	Defines the incremental distance between the arc start point and the center, measured parallel to the X and Y axes, respectively. The direction to the center is determined implicitly. Used in a G75 command, the X and Y distance can be signed ("+" to draw the circle in a positive direction, "-" to draw in a negative direction).

Polygon area fill, using G36, provides a more efficient means of filling closed polygons than stroke fill. Rather than specifying numerous G01 commands to fill an area, the G36 command allows you to simply define a polygon by its closed outline. G36 is followed by D01 commands, which are considered edges of the polygon. D02 closes and fills the polygon, and G37 turns the command off.

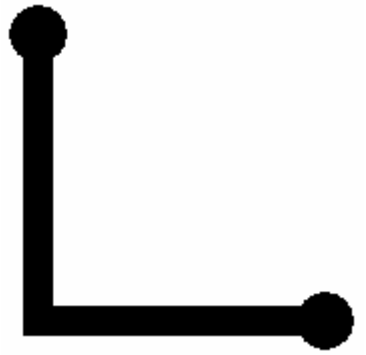
M-codes

M-codes are used to identify the end of a file.

M-code	Function
M00	Program stop
M01 or M1	Optional stop
M02 or M2	End of program

Example 274-D Files

Following is an example of a simple plot. D-code 10 is used for the round flashes on each end, and D-code 11 is used for the square draws in between.



This file was exported from VisualCAM in 274-D format, Absolute Coordinates with a Coordinate (m.n) Format of 2.4, leading zeros suppressed. The resulting 274-D Gerber file is below, with a description of each data block.

G54D10*	Prepare D-code 10.
G1X55000Y32000D3*	Flash at X:Y coordinates 5.5000:3.2000.
G1X55500Y31500D3*	Flash at 5.5500:3.1500.
G1X55000Y32000D2*	Move (shutter closed) to 5.5000:3.2000.
G54D11*	Prepare D-code 11.
G1X55000Y31500D1*	Move (draw) to 5.5000:3.1500 (this creates the vertical line segment).
G1X55500Y31500D1*	Move (draw) to 5.5500:3.1500 (this creates the horizontal line segment).
M2*	End of plot.

Each data block ends with an asterisk (*). A Gerber file can be exported in Inches or Millimeters; the one above is in Inches.

How do we know where the decimal is in the coordinates? That is where the "Coordinate Format" or "m.n" value comes into play. A 2.4 format means that there are 2 digits before the decimal point, and 4 digits after. However, this can be confusing when leading or trailing zeros are suppressed, as in the above example. When leading zeros are suppressed, then we only need to pay attention to the number to the right of the decimal point.

"Absolute Coordinates" means that the coordinates you are looking at are the actual X:Y values based on a 0:0 origin. "Incremental Coordinates" means that each X:Y coordinate is a displacement from the previous coordinate.

Following is the same Gerber file, except it was exported with no zero suppression, and in Incremental Coordinates.

```
G54D10*
G01X055000Y032000D03*
G01X000500Y-000500D03*
G01X-000500Y000500D02*
G54D11*
G01X000000Y-000500D01*
G01X000500Y000000D01*
M02*
```

The following Gerber file is an illustration of a filled polygon (the G36 command). It is a simple square.

```
G36*
G1X0Y40D02*
G75*
G3X40Y0I40J0D01*
G1X9960Y0D01*
G3X10000Y40I0J40D1*
G1X10000Y9960D1*
G3X9960Y10000I-40J0D1*
G1X40Y10000D1*
G3X0Y9960I0J-40D1*
G1X0Y40D1*
G37*
G74*
M2*
```

Your next question is likely, "How am I supposed to know what all the formatting values are?" Well, that's a disadvantage to using 274-D. Someone with a lot of experience with Gerber may be able to tell just by looking at the file, but interpreting a 274-D Gerber file can be difficult for the novice user. In a perfect world, the CAD system that created the files might actually use all the G-codes that specify the format, add comments (using the G04 command), or provide you with a separate Read Me file identifying the correct format. If you are trying to import 274-D Gerber

Files into VisualCAM and are unsure of the formatting, you should use the Import Wizard. The Import Wizard not only helps in identifying whether your files are 274-D or 274-X, but also makes an educated guess at the format of the file contents. You are provided with a preview before the files are loaded, so that you can identify any discrepancies in the format settings. If you'd rather eliminate the guesswork with file formats, use 274-X.

274-X and Apertures

The 274-X, or Extended Gerber, format has many advantages over 274-D. One advantage is that a 274-X format file does not use a separate aperture list file like the previously discussed. The aperture information is embedded in the Gerber file, along with the photoplotter commands. So we do not have to worry about converting aperture lists or having undefined apertures. Standard apertures are identified with an AD command as follows:

%ADD<D-code number><type>,<modifier>[X<modifier>]*%

Each parameter block begins and ends with a "%", and each modifier is separated by an "X". In order for VisualCAM's Import Wizard to identify a file as 274-X, at least one %ADD statement must be present. If your file does not have the statement, you will have to manually identify the Data Format as 274-X in Page 2 of the Wizard. The file will then import, but no aperture information will be present in the data.

Type	Modifier
C Circle	**<outside diameter>[X<X-axis hole dimension>[X<Y-axis hole dimension>]]** To define a solid circle, enter only the diameter. Enter one dimension to add a round hole in the center, two dimensions of equal value for a square hole. **Example:** **%ADD10C,.10X0.025X0.025*%** A .05 circle with a .025 square hole in the center.
R Rectangle/Square	**<X-axis dimension>X<Y-axis dimension[X<X-axis hole dimension>X<Y-axis hole dimension>]** To define a solid aperture, enter only the X-axis and Y-axis dimensions. A square would have dimensions of equal value. To add a hole in the center, enter one additional dimension for a round hole, two for a square/rectangular hole. **Example:** **%ADD11R,0.10X0.05X0.04X0.02*%** A .10 X .05 rectangle with a .04 X .02 rectangular hole in the center.
O Oval	**<X-axis dimension>X<Y-axis dimension>[X<X-axis hole dimension>[X<Y-axis hole dimension>]]** To define an oval that is elongated horizontally, the X dimension is larger than the Y. If the Y dimension is larger, the oval is vertical. Enter only the X and Y values for a solid aperture. Enter one dimension to add a round hole in the center, two dimensions of equal value for a square hole. **Example:** **%ADD12O,0.05X0.10*%** A horizontal, solid oval.
P Regular Polygon	**<outside dimension>X<number of sides>[X<degrees of rotation>[X<X-axis hole dimension>X<Y-axis hole dimension>]]** To define a solid polygon, enter the outside dimension and number of sides. The first point is located on the X-axis, and you can add a dimension rotate the image. Enter one additional dimension to add a round hole in the center, two dimensions of equal value for a square hole. **Example:** **%ADD13P,.10X5*%** A solid pentagon with a .10 outside dimension.

274-X also has the ability to define custom apertures, such as Targets and Donuts. AM (Aperture Macro) parameters are used define multiple shapes that are combined into a single aperture. They are then used in the same way as standard apertures.

%AM<aperture macro name>*<primitive>,<modifier$1>,<modifier$2>,[<...>]*[<primitive>[<modifiers>]]*...*%

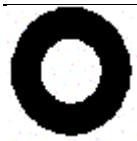
The "aperture macro name" is the name to be used in the AD parameter. Each primitive has an assigned number, as well as various modifiers. Modifiers may be absolute values, or may be variables to be supplied by the AD parameter, as described below. If an absolute value is entered instead of a variable, then the variable numbers are "shifted right". For example, if an absolute number is used for the first variable, then the next variable becomes $1.

Primitive Number	Modifier Variables
1 Circle	**$1** 0 = exposure off 1 = exposure on 2 = reverse current exposure state
	$2 Diameter
	$3 X center position
	$4 Y center position
2 or 20 Line (vector) A line whose end points are rectangular.	**$1** 0 = exposure off 1 = exposure on 2 = reverse current exposure state
	$2 Line width
	$3 X start point
	$4 Y start point
	$5 X end point
	$6 Y end point
	$7 Rotation in degrees + = counterclockwise - = clockwise
21 Line (center) A centered rectangle.	**$1** 0 = exposure off 1 = exposure on 2 = reverse current exposure state
	$2 Rectangle width
	$3 Rectangle height
	$4 X center point
	$5 Y center point
	$6 Rotation in degrees + = counterclockwise - = clockwise

Primitive Number	Modifier Variables
22 Line (lower-left) A rectangle based on a lower-left point.	**$1** 0 = exposure off 1 = exposure on 2 = reverse current exposure state
	$2 Rectangle width
	$3 Rectangle height
	$4 X lower-left point
	$5 Y lower-left point
	$6 Rotation in degrees + = counterclockwise - = clockwise
4 Outline An open or closed shape defined by a series of X:Y points (up to 50). The first and last points are the same in a closed shape.	**$1** 0 = exposure off 1 = exposure on 2 = reverse current exposure state
	$2 The number of points in the outline
	$3 X start point
	$4 Y start point
	$5 Second X point
	$6 Second Y point
	$7 Third X point
	$8, etc. Third Y point (continue as needed)
	$9, or last number used Rotation in degrees + = counterclockwise - = clockwise
5 Polygon A closed shape with sides of equal length. It is defined by 3 to 10 vertices, a center point, diameter, and rotation.	**$1** 0 = exposure off 1 = exposure on 2 = reverse current exposure state
	$2 Number of vertices
	$3 X center point
	$4 Y center point
	$5 Diameter
	$6 Rotation in degrees + = counterclockwise - = clockwise

Primitive Number	Modifier Variables
6 Moiré A cross hair centered on concentric circles.	**$1** X center point
	$2 Y center point
	$3 Outside diameter
	$4 Circle line thickness (positive data)
	$5 Gap between circles (negative data)
	$6 Number of circles
	$7 Cross hair thickness (positive data)
	$8 Cross hair length
	$9 Rotation in degrees + = counterclockwise - = clockwise
7 Thermal A solid circle of positive data, with a smaller circle and cross ties of negative data centered on it.	**$1** X center point
	$2 Y center point
	$3 Outer (positive) circle diameter
	$4 Inner (negative) circle diameter
	$5 Cross tie thickness (negative data)
	$6 Rotation in degrees + = counterclockwise - = clockwise

Custom Aperture Examples



```
%AMDONUT*
1,1,$1,0.0,0.0*
1,0,$2,0.0,0.0*
%

%ADD10DONUT,0.60X0.40%
%ADD20DONUT,0.08X0.70%
```

In the above example, D10 is defined as a 60 mil donut with a 40 mil hole and D20 is defined as an 80 mil donut with a 70 mil hole. Note that both D10 and D20 refer to the same aperture macro but have different sizes.



```
%AMTHSS3*
20,1,0.200000,-0.100000,0.000000,0.100000,0.000000,0.000000*
20,0,0.100000,-0.050000,0.000000,0.050000,0.000000,0.000000*
20,0,0.030000,0.000000,0.000000,-0.141400,0.000000,0.000000*
20,0,0.030000,0.000000,0.000000,0.070700,-0.122456,0.000000*
20,0,0.030000,0.000000,0.000000,0.070700,0.122456,0.000000*
%

%ADD64THSS3*%
```

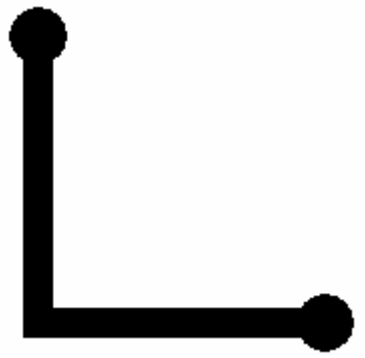
The above example assigns the aperture to D64. Absolute values are used in the Aperture Macro, so no additional values are specified in the AD command.



```
%AMOCTAGON_DIA*
5,1,8,0,0,0.09000,-22.50000*
21,0,0.04000,0.04000,0,0,45.00000*
```

Example 274-X File

If we export our example plot into a 274-X file this time (in 2.4, absolute coordinates, leading zero suppression), we get the following file:



```
*
%LPD*%
%LNLayer2*%
%FSLAX24Y24*%
%MOIN*%
%AD*%
%ADD10C,0.010000*%
%ADD11R,0.005000X0.005000*%
G54D10*
%SRX1Y1I0.0J0.0*%
G1X55000Y32000D3*
G1X55500Y31500D3*
G1X55000Y32000D2*
G54D11*
G1X55000Y31500D1*
G1X55500Y31500D1*
M2*
```

We are provided with a lot more information in this file than in 274-D. Next we will discuss each of the parameters that precede the Aperture/D-code definitions and photoplotter instructions in the example, as well as some other parameters you might see in a Gerber file.

Polarity

274-X has the concept of "Polarity", where 274-D does not. In essence, positive polarity refers to visible items. Negative polarity refers to items that clear away or erase visible items (on a plot, this reveals the background color). The LP (Layer Polarity) parameter specifies positive or negative polarity for all the data following it, until another LP

parameter is encountered. "D" stands for Dark, or positive, polarity. "C" stands for Clear, or negative polarity. In our example, the parameter is %LPD*%, or positive polarity. All items on a layer do not necessarily have to have the same polarity — the LP parameter can be used on an "item level".

Layer Name

The LN parameter assigns a name to the layer whose information follows the parameter. This name can be anything, but it usually describes either the layer's place in the stack-up, such as "Layer2", or what type of layer it is, such as "Border".

Format Statement

This is the format information we were lacking in the 274-D file. It describes zero suppression, coordinate values and coordinate format (M.N). In order for VisualCAM's Import Wizard to identify a file as 274-X, this statement must be present. If your file does not have the statement, you will have to manually identify the Data Format as 274-X in Page 2 of the Wizard. You will also have to identify the proper M.N, Coordinate Mode, and Zero Suppression in order for the file to be read properly. The file will then import.

%FS<L or T><A or I><Xn><Yn>*%

Parameter Value	Description
<L or T>	L = suppress leading zeros T = suppress trailing zeros
<A or I>	A = Absolute Coordinates I = Incremental Coordinates
<Xn> & <Yn>	n = the number of integer and decimal places for X and Y In our example 274-X file, "X24Y24" is a coordinate format of 2.4. It is customary for the values to be the same for X and Y.

MO Parameter

The MO parameter specifies if units of measure are in Inches (IN) or millimeters (MM). This is another piece of information that is omitted from 274-D (unless you used G70 or G71).

Step and Repeat

The SR parameter is used to duplicate data a specific number of times (repeats) at regular intervals (steps). When entered without parameters, it is also used to disable the previous SR parameter. Step and Repeat eliminates repetitive entry of data and helps to keep file sizes to a minimum. A common use for step and repeat is in panelization.

%SR[X<number of repeats>][Y<number of repeats>][I<X-axis step>][J<Y-axis step>]*%

Parameter Value	Description
X<number of repeats>	the number of times the data will be repeated along the X-axis
Y<number of repeats>	the number of times the data will be repeated along the Y-axis
I<X-axis step>	the distance between the X-axis repeats
J<Y-axis step>	the distance between the Y-axis repeats

Mirror Image

The MI parameter is used to "flip" data on either the X or Y axis. When turned on, all the data that follows is mirrored until another MI command is used.

The AS parameter is used to correlate the X and Y axes with the output device's A and B axes.

%MI[A<1 or 0>][B<1 or 0>}*%

Parameter Value	Description
A<1 or 0>	1 = invert A-axis data (flip image on the B-axis) 0 = disable
B<1 or 0>	1 = invert B-axis data (flip image on the A-axis) 0 = disable

The following example inverts Y-axis data, flipping the image on the X-axis.

%ASAXBY*%
%MIA0B1*%

Composites

Basically, a composite is a single layer (and hence, a single Gerber file), which is made up of a set of positive (dark) and negative (clear) layers. When you look at a composite file in ASCII format, you will see two or more layers defined,

with LP, LN, and MO parameters defined for each. They are primarily used to route traces on power planes. For more information on composites, see *Working with Composites*.

Gerber from Japan

In recent years, a version of Gerber that utilizes decimal points has emerged from Japan. In versions we have seen, instead of a leading or trailing zero (L or T) parameter in the format statement, a "D" appears. In addition, all the Gerber commands in the body of the file have the decimal points inserted in them (whereas a correct Gerber file would not have decimals, because that M.N information is gained from the XnYn parameter in the Format Statement). Although this use of decimal points does not conform to the Gerber standard, VisualCAM will import this type of Gerber file. All Gerber files we export, however, conform to the Gerber standard.

Working with Apertures

This tutorial will assist you with all aspects of working with apertures.

Importing Aperture Lists

If you are starting with 274-D Gerber data, you will have a separate Aperture List file that must be imported along with your Gerber data. (274-X files have aperture information that is embedded in the files, so no separate aperture file is needed. For a detailed explanation of what aperture lists are, see *Gerber for Beginners*.)

There is no "standard" format for aperture lists; each CAD/CAM system can use whichever format they desire. Because of this, when VisualCAM imports an aperture list, an *Automatic Conversion Rule* (ACR) file is necessary to interpret the aperture list. VisualCAM has an extensive library of aperture list converters that can be used for aperture lists from many different CAD/CAM systems. You simply select the appropriate ACR file when you use the *File|Import|Aperture List* command. If no appropriate ACR file exists, then you can interactively create your own.

If you do not import the aperture list files, you will be warned that your apertures are undefined. The VisualCAM aperture table will contain the D-code references from your Gerber file, but you will have to assign the shapes and sizes.

Regardless of the format of your database, when you have opened or imported your design data, all aperture information (if any) appears in the VisualCAM aperture table and in the Navigator. The following information will assist you in working in the aperture table that is accessed using the Setup|Apertures command. For a detailed explanation of the Aperture Setup dialog box, see the Setup|Apertures command topic.

The Apply button, under the shape tabs in the Aperture Setup dialog box, applies the currently defined values to the D-code selected in the aperture list. You can have the system automatically apply changes to selected D-codes by clicking the down-arrow button, and selecting the Auto Apply setting. For the sake of this tutorial, we will use the Apply button in its default state and not use the Auto Apply function.

Adding an Aperture

There are three methods for adding apertures to your aperture list, represented by the three aperture list view tabs on the left side of the Aperture Setup dialog box. The method you use depends upon what best suits your needs. If you are creating apertures from scratch, the D-Code View or Shape View is recommended. If you have a hardcopy aperture list that you simply need to enter the information from, the Quick Add method is recommended.

In D-Code View

1. Click the Add D-Code button.
2. The next available D-code number appears in the New D-Code dialog box. Accept the number, or type in a different, unused D-code number. Click OK, and an undefined aperture appears in the list.

Tip: You can also add multiple new D-codes by using the following convention: ***n,n*** *adds D-codes using the numbers you specify (****12,14*** *adds D-codes 12 and 14), and* ***n:n*** *adds a range of D-codes (****12:14*** *adds D-codes 12, 13, and 14).*

3. Select your desired aperture shape by clicking on one of the shape tabs in the middle of the dialog box.
4. Define the parameters, as necessary. See the Setup|Apertures command topic for a detailed explanation of each aperture type.

   Note that a representation of your aperture appears in a preview on the right side of the dialog box. Although it is not an exact representation of the size of the aperture, it shows you how the aperture shape will appear.

   Custom apertures are defined using the Custom Aperture Editor. To create a new custom aperture, click the Custom Ap button, and select the New command from the menu. To use a pre-existing custom aperture from a library file, click the Custom Ap button, and select the Load From Lib command from the menu. Once you have created the custom aperture or loaded the library file, it appears in the Custom Aperture Name list. Select the custom aperture from that list to use it as a D-code.
5. When you are finished, click the Apply button. Your aperture is now defined.
6. If your D-code requires an angle of rotation, click on the Angle column for the D-code, and the Angle is immediately highlighted. Type in the new value.
7. To define an aperture's type (SMT, Thermal, or ThruHole), click on the Type column for the D-code, and select the desired type.

In Shape View

1. Right-click on the desired aperture shape, and select Add from the shortcut menu.

   -or-

   Click on the desired aperture shape, and click the Add D-Code button.

2. The next available D-code number appears in the New D-Code dialog box. Accept the number, or type in a different, unused D-code number. Click OK, and an undefined aperture appears in the list.
3. Define the parameters, as necessary, under the shape tab in the middle of the dialog box. See the Setup|Apertures command topic for a detailed explanation of each aperture type.

   Note that a representation of your aperture appears in a preview on the right side of the dialog box. Although it is not an exact representation of the size of the aperture, it shows you how the aperture shape will appear.

   Custom apertures are defined using the Custom Aperture Editor. To create a new custom aperture, click the Custom Ap button, and select the New command from the menu. To use a pre-existing custom aperture from a library file, click the Custom Ap button, and select the Load From Lib command from the menu. Once you have created the custom aperture or loaded the library file, it appears in the Custom Aperture Name list. Select the custom aperture from that list to use it as a D-code.
4. When you are finished, click the Apply button. Your aperture is now defined.
5. If your D-code requires an angle of rotation, double-click on the D-code number to expand the tree view. If an angle of rotation can be applied to the D-code, a "0" appears underneath the D-code number. Right-click on the "0", and a shortcut menu appears. Select the Edit command, and type in the new value.
6. The aperture type also appears under the D-code number. To define an aperture's type (SMT, Thermal, or ThruHole), right-click on the existing type ("SMT", by default), and a shortcut menu appears. Select the desired type.

In Quick Add

The Quick Add function is used to create basic apertures. You should view your individual D-codes in the D-code or Shape view to verify that they are correct, and select the appropriate aperture type (SMT, Thermal, or ThruHole). Some apertures that can have complex definitions, such as Thermals and Moires, may need to be edited.

Custom apertures cannot be defined using the Quick Add command.

1. Type your aperture information, in the specified format, into the Quick Add text box. Place the information for each D-code on its own line.
2. When you are finished, click the Apply button under the text box. Your aperture information has been added to the table, and will appear in the Shape and D-code views.

Editing an Aperture

All your aperture definitions can be edited in either the D-code or Shape views. If you need to change a D-code number, you must delete the existing D-code, and then add a new one with the correct number.

Tip: To revert back to the previously saved parameters at any time (even after the changes have been applied), click the Reset button.

In D-Code View

1. Select the desired aperture by clicking on it in the Properties column. The appropriate Shape tab, in the middle of the dialog box, is automatically brought into focus.
2. Change your aperture definitions, as necessary.

   To change the angle of rotation for a D-code, click on the Angle column for the D-code, and the Angle is immediately highlighted. Type in the new value. To change an aperture's type (SMT, Thermal, or ThruHole), click on the Type column for the D-code, and select the desired type.
3. When you are finished, click the Apply button. The definitions are updated in your aperture list.

In Shape View

1. Expand the tree view by double-clicking on the desired aperture's shape category, and then on the desired D-code number. The appropriate Shape tab, in the middle of the dialog box, is automatically brought into focus.
2. Change your aperture definitions, as necessary, and click the Apply button.
3. To change the angle of rotation, right-click on the existing angle underneath the D-code number, and a shortcut menu appears. Select the Edit command, and type in the new value.
4. To define an aperture's type (SMT, Thermal, or ThruHole), right-click on the existing type, and a shortcut menu appears. Select the desired type.

Deleting an Aperture

Only unused D-codes can be deleted.

Tip: The Compact button removes unused and redundant apertures within an aperture list. Each layer then has its D-codes remapped accordingly.

In D-Code View

1. Right-click on the desired aperture in the Properties column.
2. Select the Delete D-code command from the shortcut menu. The D-code is immediately deleted.

In Shape View

1. To delete an individual D-code, expand the tree view for the desired aperture shape by double-clicking on it. Then double-click on the desired D-code size.
2. Right-click on the desired D-code number.
3. Select the Delete command from the shortcut menu. The D-code is immediately deleted.

   -or-

1. To delete all D-codes for a particular aperture definition (for example, all 4 mil Rounds), expand the tree view for the desired aperture shape. Right-click on the desired D-code size.
2. Select the Delete command from the shortcut menu. All the D-codes listed under that size are deleted.

Exporting Aperture Lists

The File|Export|Aperture List command exports an aperture list file in the VisualCAM .map format. Exporting this file is only necessary if you are exporting your design data in the Gerber 274-D format.

When you export your data in other file formats, or save a .vcam file, the aperture information is contained within those files. You do not have to save a separate aperture list file.

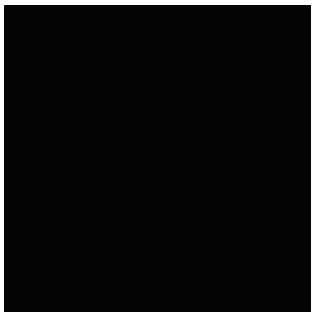
Working with Composites

The following tutorials explain the most common tasks involved in working with composites:

What is a Composite?

A composite a single layer (and hence, a single Gerber file), which is made up of a set of positive (dark) and negative (clear) layers. They are used to route traces on power planes. In the example below, the first layer is dark, a second clear layer is used to create a path for the trace, and then a dark trace layer is used to make the connections. (For display purposes, the clear data element below is shown in black.)

Original Copper Area (Dark Layer):



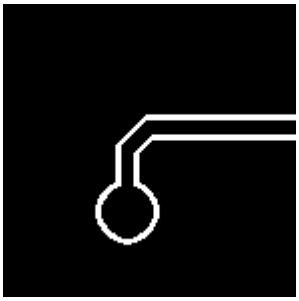
Oversized Data (Clear Layer):



Trace (Dark Layer):



Resulting Composite:



By allowing the use of black and white for layer colors, VisualCAM allows accurate viewing of composite power and ground layers. Setting the negative layer to white and the positive layers to black will result in a very realistic depiction of the final film.

Tip: For accurate viewing and editing, it is important that the layers be placed in the proper order in the layer list. Also make sure the appropriate layer is active. In the above three-layer example, if the first dark layer is active, you will not see the other two composite layers.

The layers that are contained in composites may be edited as you would any other layers, using the standard VisualCAM Edit commands.

Because VisualCAM supports item-level polarity (items of different polarity can exist on a single, standard layer), creating composites is not necessary. However, as composites are still commonly used, the following tutorials explain how to work with them in VisualCAM. In many instances, both the Navigator and standard menu commands can be used to view and manipulate composites. Both methods are explained, so that you may choose whichever works best for you.

Warning: The standard 274-D format does not support composite layers or item-level polarity; to properly export a composite or layers that contain a combination of positive and negative data, you must use 274-X.

Importing a Composite

1. Select the File|Import|Import Wizard command.
2. In the first dialog box, select the directory that your composite file is in.
3. In the second dialog box, select the composite file you wish to import, and unselect any other files you do not wish to import. There is no need to load in an aperture list, as it is included in the 274-X file.

4. Click in the Data Format column. The Gerber Import Data Format dialog box appears.
5. By default, VisualCAM converts composites to single layers with item-level polarity. This allows for easier editing of the data. To maintain your data as a standard composite, select the Layer Composites option (under Assign 274-X Polarity To).
6. Click OK and complete the import process.

Creating a Composite

Creating a New Composite

If the layer you wish to add to the composite does not already exist, select the Setup|Layers command. Find a new, empty layer in the Layer Setup dialog box, and assign it a name and an appropriate layer type.

1. Select the Setup|Composites command. The Composite Setup dialog box appears.
2. The Composite List shows any currently loaded composites and the layers contained within them. Click the Create Composite button and a new composite with a system default name (such as "Composite 1") is created.
3. To add a layer to a composite, click and drag the desired layer from the Layer List to the Composite List. To remove a layer from a composite, drag the layer back to the layer list.
4. To change layer polarity, right-click on a layer within a composite to select a different polarity. A polarity of Dark means that the layer is to be displayed in the style a normal Gerber file is displayed. Clear tells VisualCAM to display the layer using the current background color. This has the effect of erasing, or "clearing", areas from an image that were previously drawn by a "dark" layer. Negative layers should be set to clear.

Using the Navigator to Create a Composite

If the layers you wish to add to the composite do not already exist, right-click on the Layers heading in the Navigator. A shortcut menu appears. Select the Add Layer command, and a dialog box prompts you to enter the new layer's name. The new layer appears in the list. It is automatically given the type "Other" and its visibility is turned off.

1. In the Navigator, right-click on the Composites heading. A shortcut menu appears.
2. Select the Add Composites command. A new composite with a system-default name appears in the Composites list.
3. Click on a layer in the Layers list.
4. While still holding down your mouse button, move the cursor to the desired composite, and release the button (i.e. drag-and-drop the layer into the composite).

   Note that if a layer already resides in a composite, it cannot also be added to another composite. To remove a layer from a composite, right-click on the desired layer, and a shortcut menu appears. Select the Remove command, and the layer is removed from the composite. This does not delete the actual layer in the database—it just removes the layer from the composite. To delete layers, see the Layers area of the Navigator.

Converting a Set of Gerber 274-D Files into a Composite File

1. Select the File|Import|Import Wizard command to import the 274-D files.
2. Select the Setup|Composites command. The Composite Setup dialog box appears.
3. The Composite List shows any currently loaded composites and the layers contained within them. Click the Create Composite button and a new composite with a system default name (such as "Composite 1") is created.
4. To add a layer to a composite, click and drag the desired layer from the Layer List to the Composite List. To remove a layer from a composite, drag the layer back to the layer list.
5. To change layer polarity, right-click on a layer within a composite to select a different polarity. A polarity of Dark means that the layer is to be displayed in the style a normal Gerber file is displayed. Clear tells VisualCAM to display the layer using the current background color. This has the effect of erasing, or "clearing", areas from an image that were previously drawn by a "dark" layer. Negative layers should be set to clear.

⚠ *Warning: If you choose to export your composite file in a Gerber Format, you must use 274-X.*

Viewing a Composite

Composite layers may be displayed WYSIWYG by simply pressing the V hotkey. This nested command toggles composite viewing on/off. Composite viewing may also be controlled using the Setup|Composites command.

Using the Navigator to View a Composite

1. In the Navigator, right-click on the desired composite. A shortcut menu appears, a checkmark next to the Visible command indicates if the layers are visible.
2. If a checkmark does not appear, select the Visible command, and the layers in the composite are now visible in the workspace.

To hide the layers, select the Visible command again.

Editing a Composite

Deleting a Composite

1. Select the Setup|Composites command. The Composite Setup dialog box appears.
2. In the Composite List, select the composite you wish to delete.
3. Click the Delete Composite button. This command does NOT delete any data or remove any data layers. Only the association to other layers is removed. To delete layers, use the Setup|Layers command or the Navigator.

Using the Navigator to Delete a Composite

1. In the Navigator, right-click on the desired composite in the list. A shortcut menu appears.
2. Select the Remove command, and the composite is immediately deleted. This does not delete the actual layers in the database—just the composite information. To delete layers, see the Layers area of the Navigator.

   -or-

1. To delete an empty composite, click on the desired composite in the list.
2. Press the DEL key, and the empty composite is immediately deleted.

Tip: If you delete a composite layer set, the layers will remain tagged as "Composite". You will need to tag your layers with a new layer type.

Adding a Layer to a Composite

If the layer you wish to add to the composite does not already exist, select the Setup|Layers command. Find a new, empty layer in the Layer Setup dialog box, and assign it a name and an appropriate layer type.

1. Select the Setup|Composites command. The Composite Setup dialog box appears.
2. To add a layer to a composite, click and drag the desired layer from the Layer List to the Composite List. Note that if a layer already resides in a composite, it cannot also be added to another composite.
3. Click OK to close the dialog box.

Using the Navigator to Add a Layer to a Composite

If the layer you wish to add to the composite does not already exist, right-click on the Layers heading in the Navigator. A shortcut menu appears. Select the Add Layer command, and a dialog box prompts you to enter the new layer's name. The new layer appears in the list. It is automatically given the type "Other" and its visibility is turned off.

1. In the Navigator, click on the desired layer in the Layers list.
2. While still holding down your mouse button, move the cursor to the desired composite, and release the button (i.e. drag-and-drop the layer into the composite).

   Note that if a layer already resides in a composite, it cannot also be added to another composite.

Removing a Layer from a Composite

1. Select the Setup|Composites command. The Composite Setup dialog box appears.
2. The Composite List shows any currently loaded composites and the layers contained within them. To remove a layer from a composite, click and drag the layer from the Composite List to the Layer List.
3. Click OK to close the dialog box.

   This does not delete the actual layer in the database—it just removes the layer from the composite. To delete layers, use the Setup|Layers command or the Navigator.

Using the Navigator to Remove a Layer from a Composite

1. In the Navigator, click on the plus box next to the desired composite. A list of layers in the composite appears.
2. Right-click on the desired layer, and a shortcut menu appears.
3. Select the Remove command, and the layer is removed from the composite. This does not delete the actual layer in the database—it just removes the layer from the composite. To delete layers, see the Layers area of the Navigator.

Converting (Flattening) a Composite to a Single Layer

- Select the Tools|Convert|Composite To Layer command. Select the source composite, and the empty destination layer in the dialog box, and click OK

  -or-

In the Navigator, right-click on the desired composite in the composite list, and select the Convert To Layer command from the shortcut menu. The composite is immediately converted, and is placed on the first available empty layer.

Exporting a Composite

1. Select the File|Export|Gerber command.
2. While selecting the desired settings in the Export Gerber dialog box, click the Format button, and change the Dialect selection to 274-X.

Plotting a composite layer on a printer is equally simple. Just view your composite layers as described above and then use the File|Print command. The plotted image will appear on the page exactly as it does on the display.

Tip: Since the image for printing is created in a high resolution off screen bitmap, the film box and display grid may appear on the output page. You can disable this by setting the film box color to the background color using the Options|Configure command, and disabling the display of the grid using the G hotkey.

Performing DRC & DFF Analyses

The VisualCAM Analysis functions are used to find any flaws in your design prior to manufacturing. Using rules that you specify, the system analyzes your design, finds errors, and determines the best fix for those errors. You can then review the errors in the workspace, print error reports, fix the errors yourself or have the system fix the errors for you.

For maximum efficiency, all your rules can be saved for use in future designs, and each check can be run at the same time.

You can also perform a graphical netlist comparison, using an IPC-D-356 or 356A netlist that you import, and the netlist extracted from your design in VisualCAM. Any discrepancies are displayed to you in a report and in the workspace, and you can then modify your design data as necessary. The Netlist Compare function is available to you both as a separate menu command, and in the DRC/MRC dialog box so that you can run the comparison at the same time as your other DRC/MRC checks.

Prior to running the Analysis functions, some data preparation is required. The preliminary steps are included below, and you can click on the hyperlinks for more details on each function.

Performing Analyses

1. Import your data or open a previously-created VisualCAM .vcam file.
2. Using the Setup|Layers command or the Navigator, make sure that each layer is "tagged" with the correct layer type. This is required so that the system understands what each layer is.
3. Remove any extraneous data on your electrical layers that are outside the perimeter of the board, such as title blocks, targets, crop marks, etc.

We recommend that any documentation for the board be done on a separate Drawing layer. If you wish to keep the data, placing it on a drawing layer assures it will not interfere with any processing, such as netlist extraction.

4. If you have any drawn pads in your database, select one of the Tools|Convert|Drawn Pads commands to convert them to flashes.
5. If any layers are misaligned, use the Edit|Align Layers command to align them.
6. Use the Tools|Pad Removal|Stacked command to remove any redundant pads.
7. Extract a netlist from your data using the Tools|Netlist|Generate command.
8. If you plan to run the Netlist Compare (NLC) analysis, import your CAD system (external) netlist using the File|Import|Netlist|IPC-D-356 command.
9. Select the Analysis|DRC/MRC command and run your analysis.
10. After performing the analysis, VisualCAM automatically displays the errors in the Navigator, which you may use to view and automatically fix the errors.

Related topics

Comparing Netlists Tutorial
Analysis|Find Duplicates command

Comparing Netlists

Graphical netlist comparison allows you to visually compare your design data against the original netlist. VisualCAM performs the comparison automatically, and all violations are stored internally like DRC/MRC and DFF violations. This means you can quickly scan through and fix connectivity issues.

As an aid to those who wish to compare a netlist created from Gerber data back to an ODB++ or PADS ASCII database netlist, the File|Import|Netlist|ODB++ and File|Import|Netlist|PADS ASCII commands import only the netlists from those databases. It is not necessary to use the File|Import|ODB++ or File|Import|PADS ASCII command to import the entire database, export an IPC-D-356 netlist, discard the database, load your Gerber data and import the IPC-D-356 netlist.

Comparing Netlists

1. Use File|Import|Import Wizard to import your Gerber data or File|Open to a previously-created VisualCAM .vcam file.

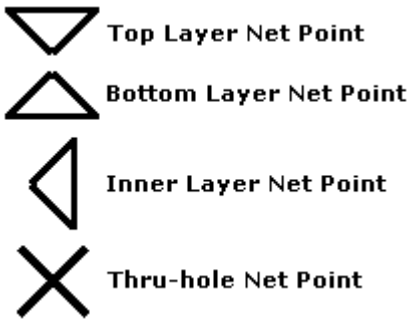
Tip: If you are importing an ODB++ or PADS ASCII database for the purposes of verifying their netlists against the artwork (not Gerber data), you can automatically run a netlist comparison when importing those databases. See the File|Import|ODB++ and File|Import|PADS ASCII topics for details.

2. Using the Setup|Layers command or the Navigator, make sure that each layer is "tagged" with the correct layer type. This is required so that the system understands what each layer is.
3. Remove any extraneous data on your electrical layers that are outside the perimeter of the board, such as title blocks, targets, crop marks, etc.

Tip: We recommend that any documentation for the board be done on a separate Drawing layer. If you wish to keep the data, placing it on a drawing layer assures it will not interfere with any processing, such as netlist extraction.

4. If you have any drawn pads in your database, select one of the Tools|Convert|Drawn Pads commands to convert them to flashes.
5. Use the Analysis|DRC/MRC command to make sure your layers are properly aligned. If any layers are misaligned, you can use the Edit|Align Layers command to align them.
6. Use the Tools|Pad Removal|Stacked command to remove any redundant pads.
7. Select the Tools|Netlist|Generate command, or use the Navigator shortcut command, to extract a netlist from your data.
8. Import your CAD system (external) netlist using the File|Import|Netlist|IPC-D-356 command, File|Import|Netlist|ODB++ command, or File|Import|Netlist|PADS ASCII command, depending upon where your "golden" netlist resides.

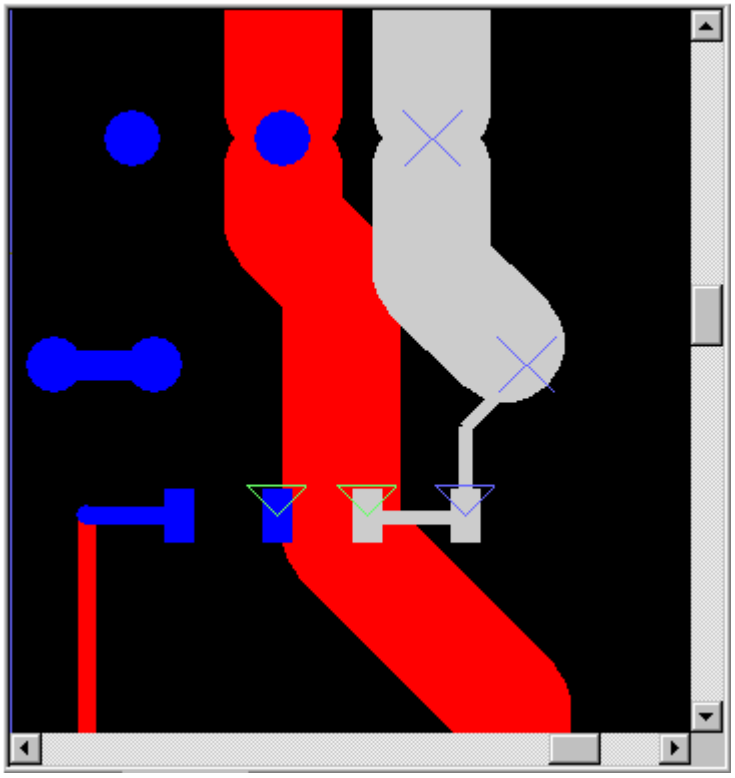
   After you import the netlist, a new External Net layer is created. Viewed by itself, this layer appears to contain no data. However, this layer contains the external netlist information, and links the external net points to the appropriate top/inner/bottom layer net test points. To view the points, turn on the associated top, bottom, and inner layers, and the external net layer. The net points are displayed in four ways for easy reference:



9. Select the Analysis|Netlist Compare command.
10. After performing the analysis, discrepancies are automatically displayed in the Navigator, which you may use to view the errors in the workspace and print reports.
11. Double-click on the error in the Navigator to view it in the workspace. The VisualCAM internal net in question highlighted, and zoomed to fit the screen. The external net points are shown in a unique color for each net.

Tip: The external net point graphic colors are arbitrary. If necessary, you can change the highlight, draw, and flash colors for the layers you are looking at for better contrast and easier visibility.

In the below example, a Net Short was detected. The blue net point symbols represent one external net (Net X), and the green net point symbols represent another (Net Y). Note that one of Net Y's points lies on the same VisualCAM net as Net X. A trace was mis-routed.



If the error is an open or a short, you can find the location of the error more quickly by right-clicking on the error in the Navigator, and selecting the Pin-Point Error command. The system then places a bounding-box around the area where the open or short is located.

You can now change the data as necessary. Once you have done so, you should then re-run the Netlist Compare command to verify that the fix was accurate. Once no more errors exist, you can use the Tools|Netlist|Apply ExternalNet Names command to apply the external net names to the database, if you wish.

Panelization

Panelization is the process of creating an array of your PCB for a manufacturing panel. This allows multiple PCBs to be created at one time, optimizing material usage and shortening your manufacturing turnaround time.

VisualCAM's Advanced Panelization allows you to panelize single or multiple designs, and lets you store an unlimited number of panel templates so you can use the same panel configuration for multiple jobs. You must have each design already saved as a VisualCAM .vcam database (this command does not automatically operate on the design currently loaded in the workspace). Coupon placement, venting and thieving, pinning holes, and title blocks are supported, as long as those elements have been saved as individual .vcam files.

Essentially, all panel elements are just seen as individual images by VisualCAM, that you can add, rotate, and repeat however you wish. As long as you prepare your data properly, panelization is a quick and easy process.

Preparing Your Data

The following steps should be followed for any images you wish to appear on the board, including coupons, pinning holes, title blocks, etc.

1. Before panelization, import or open your data.
2. Using the Setup|Layers command or the Navigator, make sure that each layer is "tagged" with the correct layer type. Layer types are very important in the panelization process. Non-electrical layers, such as drawing layers, will not be included in the finished panel.
3. Remove any extraneous data on your electrical layers that are outside the perimeter of the board, such as title blocks, targets, crop marks, etc.
4. If necessary, create a mill path around your image and optimize your NC tooling.
5. Complete any other data preparation and analysis that is necessary prior to sending your job to manufacturing.
6. Save your data as a .vcam file.

Creating a New Panel

1. Prepare the images you wish to panelize as instructed above.
2. Select the File|New command to start with an empty database.
3. Select the Tools|Panelize|Advanced command, and define the various elements of your panel.
4. When you click OK in the Advanced Panelization dialog box, your finished panel appears in the workspace.
5. Save this database as a .vcam file, or export it in whatever format you require.

The panel is created with step and repeat codes, so you cannot edit the individual "virtual" images. You can, however, change the placement of the virtual images within the panel. We recommend, if you wish to modify the image placement the panel, that you select the Tools|Panelize|Advanced command again and modify the panel there. That will prevent a violation of any spacing rules you set up, and any venting pattern can be adjusted automatically.

If you save your panel as a .vcam file, and then change *any* aspect of your original design (in the original .vcam file you started with), the next time you open your panel file you will be warned that your data has changed. You should then use the Advanced Panelization dialog box to verify that the layout and spacing rules are still applicable to your modified design, and that the layer sets are still correct.

Panelizing Multiple Images

1. If you are including multiple designs on the same panel, you first need to save each design as a .vcam file. See Preparing Your Data above.
2. Under the Image tab of the Advanced Panelization dialog box, browse for as many images as you would like to include in the panel. The file that appears in the Image list establishes which image you are working with at the moment, and you simply select a different image to define its panel properties.
3. You do not need to worry about the layer order in the original .vcam files. The first file you bring into the panel establishes an initial panel layer structure. As you add new images to the panel, you can then use the Layer Mapping function to match each image layer to the appropriate panel layer, and create new panel layers as necessary. So you can make sure that all image layer types on each panel layer match.
4. When laying out multiple images on a panel, you must use the Manual Placement option under the Layout tab. You can then specify the exact number of copies you want on the panel, then move and rotate them individually to optimize board space. To change the placement, right-click on the image you want to move, and a shortcut menu appears. You can Move, Rotate, or Delete images at will.

Adding Panel Elements

Adding Panel Elements functions the same as panelizing multiple images.

1. Each element, such as a coupon, title block, or fiducial, must be individually saved as a .vcam file. When creating the element, you should tag it with the layer type it will appear on (only electrical layers are included in a panel).
2. Under the Image tab of the Advanced Panelization dialog box, you can then browse for whatever elements you would like to include in the panel. The file that appears in the Image list establishes which image you are working with at the moment, and you simply select a different image to define its panel properties.
3. As you add new images to the panel, use the Layer Mapping function to define which panel layer you want the element to appear on. You must then use the Manual Placement option under the Layout tab to place the item on the panel. You can then specify the number of copies of the element you want on the panel, then move and rotate them individually to place them. To change the placement, right-click on the image you want to move, and a shortcut menu appears. You can Move, Rotate, or Delete images at will.

Venting, Thieving, and Robber Bars

Venting & thieving, and robber bars are each defined under their own respective tabs in the Advanced Panelization dialog box. Please see the Advanced Panelization command topic for an explanation of the definitions.

Venting is normally applied to inner layers. Inner layers usually have less copper area than outer layers, because there are usually fewer traces, etc. Venting equalizes the amount of copper area on the inner layers, so that the amount of plating material applied during manufacturing is not only the same amount, but the same thickness. Otherwise, the inner layers become thicker than the outer layers, because they have the same amount of plating material applied to a smaller area of copper.

Venting is also important during the lamination process. Cross-hatch and dot patterns help the preg material flow out while the layers are pressed together, creating an even distribution of preg material and preventing bubbles from forming between the layers. It is also helpful if the patterns on each layer are offset from each other, so that the cross-hatches or dots on the layers fit between each other when pressed together, forming a snug fit.

Robber bars, or galvanic vents, are often added to top and bottom layers to balance the copper between them, to prevent over-etching. (Use Copper Area analysis to see if this is necessary for your design). They can also be used as a grounding strip for plating.

Creating a Panel Template

Many manufacturers have standard panel attributes that are used for several jobs. You can create pre-defined panel "templates" for use on multiple jobs, to streamline the panel creation process. There are several pre-defined panel templates and elements provided in the VisualCAM program directory, in the Samples/Template folder.

1. Define a panel as described above, with whatever attributes you wish (size, venting, panel elements, layer setup, etc.).
2. If you have a standard tool table, use the Setup|NC Tools to define your shop's tooling. If you do not define a tool table in your template, the first image that you load into your template upon panelization will determine the tool table definitions.
3. Save the template as a .vcam file.
4. To use the template in the future, use the Advanced Panelization function and load the saved panel template image first.
5. Add whatever images you want to step & repeat, and define their placement, layer mapping, etc. in relationship to your panel template.
6. When you define the layer mapping, if any NC data exists in the images, you are also required to map tool definitions to your template's tool table. Again, if you did not define a tool table in your template, the first image you load into the template will define the tool table. If you load additional images into the table, you must then map their tools based upon the definitions from the first image.
7. When you have completed your panel, save it as a new .vcam file or export it as necessary. By saving your completed panel as a new file, your template file remains untouched, and you have a new panel ready for manufacturing.

Creating Test Fixture Files

VisualCAM supports two types of bare board test: bed of nails and flying probe. Tutorials for generating the necessary test files for each type of test are below.

Recommended reading on the basics of bare board testing:

"Bare Board Electrical Testing" by Philip J. Hallee, Everett Charles Technologies

"The Continued Evolution of Test Data Transfer Format IPC-D-356" by Duane Delfosse, The Board Authority

Bed of Nails Test Procedure

Bed of nails testing is also referred to as "fixtured" or "clamshell" testing. Probes or pins, arrayed on a grid, are used to make contact with each point on a circuit to determine if the connection is good. To accomplish this, a netlist is used to determine where the probes must make contact with the board. Special plates are used to guide the probes to the desired points, so these plates must be defined and drilled. The following tutorial explains how to prepare your data, and create the necessary files for testing.

1. Import or open your data.
2. Using the Setup|Layers command or the Navigator, make sure that each layer is "tagged" with the correct layer type. This is required so that the system understands what each layer is.
3. When you create your test points, they will be added to a new layer. The layers should be named appropriately.

   If you selected the Setup|Layers command, find an empty, unnamed layer in the Layer Setup dialog box. If you will be testing only the top side of your board, name the empty layer **test_top**. If you will be testing only the bottom side, name the empty layer **test_bot**. If you are testing both sides of your board, you will need to select two empty layers and name them **test_top** and **test_bot**, respectively. The other information for the layer should remain the default setting.

   If you are working in the Navigator, right-click on the Layers heading, and select Add Layer from the shortcut menu. If you will be testing only the top side of your board, name the empty layer **test_top**. If you will be testing only the bottom side, name the empty layer **test_bot**. If you are testing both sides of your board, you will need to select the Add Layers command twice to create two empty layers. and name them **test_top** and **test_bot**, respectively.
4. Remove any extraneous data on your electrical layers that are outside the perimeter of the board, such as title blocks, targets, crop marks, etc.

Tip: We recommend that any documentation for the board be done on a separate Drawing layer. If you wish to keep the data, placing it on a drawing layer assures it will not interfere with any processing, such as netlist extraction.

5. If you have any drawn pads in your database, select one of the Tools|Convert|Drawn Pads commands to convert them to flashes.
6. Use the Analysis|DRC/MRC command to make sure your layers are properly aligned. If any layers are misaligned, you can use the Edit|Align Layers command to align them.
7. Use the Tools|Pad Removal|Stacked command to remove any redundant pads.
8. Select the Tools|Netlist|Generate command to extract a netlist from your data.
9. Select the Tools|Test Points|Setup command. Select the Bed Of Nails option, and indicate whether you are testing a single- or double-sided board.

   If you are testing a single-sided board, indicate whether the tested (Output) layer is the Top or Bottom layer. The Target Layer is the layer that you wish to place your test point graphics on. Select the appropriate layer, which you created in Step 3.

   If you are testing a double-sided board, select the appropriate Target Layer for each side of the board. **test_top** should be the top target layer, **test_bot** should be the bottom target layer.
10. Now you need to select the parameters for your fixture plates and probes. In the Grid Size box, select the spacing between grid points (this is a single-density grid).
11. Click the Probe Setup button. The Probe Setup Table contains a list of probes assigned to pads that are created using the D-codes given in the list. By default, the probe with code 1 is assigned to all pads. To change the assignment, click on the desired Probe Code and type the new number in the text box.

    To change the Probe Code settings, click on the Probe Properties button. The Probe Properties dialog box appears. The probe diameter, length and maximum deflection angle (the largest angle a probe can employ while testing a point) appear below the table. To change any values, click on any item in the list and type the new value. When you are finished defining your probes, click OK. Click OK again in the Test Setup dialog box.

    VisualCAM will use the Fixture Parameters when generating the fixture plate drill files for this design. If a tool of the exact size required is not found, the closest larger size tool will be used.

12. Select the Tools|Test Points|Generate command. Now you are specifying what types of pads are tested. Ends Only specifies that only the ends of traces are to be tested. All specifies that all pads are to be tested. Select Ignore Isolated to overlook isolated pads. Click OK when you have completed your selections.

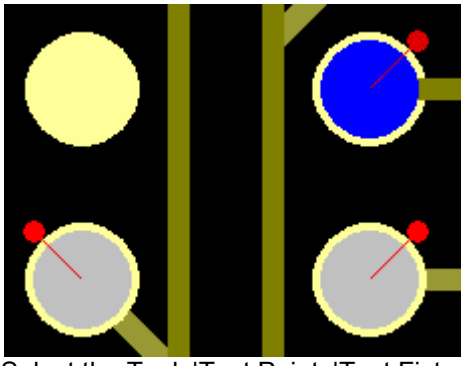
    The system scans your data and creates the test points. The graphics representing your test points appear on the test_top and/or test_bot layers.

13. You can add additional test points, if necessary, using the Tools|Test Points|Add command. To delete test points, either select the Edit|Undo command immediately after generating test points, or use the Edit|Delete command and set the selection filter to delete only flashes on the test_top or test_bot layer where the test points are placed.

14. Automatic test point staggering is used to provide additional spacing between test probes, to insure there is no probe-to-probe contact. It does this by offsetting the location of the test point from the center of the corresponding pad. Staggering is necessary if you have a large number of probes in a small area, such as a fine-pitch SMT component. Select the Tools|Test Points|Stagger command if you wish to stagger any test points.

15. Select the Tools|Test Points|Test Fixture|Snap To Grid command to "snap" the test points to the fixture plate grid. This tells VisualCAM where to create the drill holes in the fixture plates.

    The following graphic shows an example of what should appear in the workspace, at this point (the colors of the items may differ from your workspace). The blue and gray flashes on the test_top and test_bot layers represent test points. The red and orange flashes show where the probes initially enter the fixture guide plates, and the line shows the path of the probe from the plate to the test point.



16. Select the Tools|Test Points|Test Fixture|Save Files command. This is where you specify all the parameters defining the probes and fixture plate requirements. You can define up to seven fixture plates. The Grid plate is the plate closest to the fixture machine, and the Board plate is the plate closest to the PCB. The Guide plates are aids for guiding the probes to the correct location on the board.

Tip: If you are creating a double-sided text fixture, you must specify the file information for both sides of the board. You are first presented with the information for the top side. To specify the information for the bottom side, click the Bottom Side button at the bottom of the dialog box. The button is a toggle, so you can switch from top to bottom side at will.

17. Select (check) the number of guides you require, and change the default names, if you wish.

18. Dist is the distance of each plate to the board. Thick is the thickness of the fixture plate. You can change either of these values.

19. Enter the desired netlist filename. By default, the netlist file is saved in the native VisualCAM .nl format. To save the netlist as an IPC-D-356A file (.net), select the IPC-D-356A option.

20. The Grid/Board Map File is an ASCII report file that shows the probe assignment for each location, the coordinates of the test point, and corresponding probe grid location. Specify a name for the .txt file.

21. Click the NC Format button if you wish to change the format used for the output test point drill file.

22. If you wish to change any of the tool information in the NC Tool table, click the Tool Setup button.

23. When you are finished defining the fixture information, click OK to save the files.

Flying Probe Test Procedure

Flying probe is also known as "moving probe" and "X:Y prober". Unlike bed of nails testers, which test multiple nets at once, flying probe test systems use two or more points which rapidly move around the board area, testing each net individually. Due to the nature of the testing, it is not subjected to limitations due to test point density. No staggering or guide plates are required to aid the test machine.

1. Import your data or open a previously-created VisualCAM .vcam file.
2. Using the Setup|Layers command or the Navigator, make sure that each layer is "tagged" with the correct layer type. This is required so that the system understands what each layer is.
3. When you create your test points, they will be added to a new layer. The layers should be named appropriately.

If you selected the Setup|Layers command, find an empty, unnamed layer in the Layer Setup dialog box. If you will be testing only the top side of your board, name the empty layer **test_top**. If you will be testing only the bottom side, name the empty layer **test_bot**. If you are testing both sides of your board, you will need to select two empty layers and name them **test_top** and **test_bot**, respectively. The other information for the layer should remain the default setting.

If you are working in the Navigator, right-click on the Layers heading, and select Add Layer from the shortcut menu. If you will be testing only the top side of your board, name the empty layer **test_top**. If you will be testing only the bottom side, name the empty layer **test_bot**. If you are testing both sides of your board, you will need to select the Add Layers command twice to create two empty layers. and name them **test_top** and **test_bot**, respectively.

4. Remove any extraneous data on your electrical layers that are outside the perimeter of the board, such as title blocks, targets, crop marks, etc.

Tip: We recommend that any documentation for the board be done on a separate drawing layer. If you wish to keep the data, placing it on a drawing layer assures it will not interfere with any processing, such as netlist extraction.

5. If you have any drawn pads in your database, select one of the Tools|Convert|Drawn Pads commands to convert them to flashes.
6. Use the Analysis|DRC/MRC command to make sure your layers are properly aligned. If any layers are misaligned, you can use the Edit|Align Layers command to align them.
7. Use the Tools|Pad Removal|Stacked command to remove any redundant pads.
8. Select the Tools|Netlist|Generate command to extract a netlist from your data.
9. Select the Tools|Test Points|Setup command. Select the Flying Probe option, and indicate whether you are testing a single- or double-sided board.
10. If you are testing a single-sided board, indicate whether the tested (Output) layer is the Top or Bottom layer. The Target Layer is the layer that you wish to place your test point graphics on. Select the appropriate layer, which you created in Step 3.

    If you are testing a double-sided board, select the appropriate Target Layer for each side of the board. **test_top** should be the top target layer, **test_bot** should be the bottom target layer.
11. Click OK to close the Test Setup dialog box.
12. Select the Tools|Test Points|Generate command. Now you are specifying what types of pads are tested. Ends Only specifies that only the ends of traces are to be tested. All specifies that all pads are to be tested. Select Ignore Isolated to overlook isolated pads. Click OK when you have completed your selections.

    The system scans your data and creates the test points. The graphics (flashes) representing your test points appear on the test_top and/or test_bot layers.
13. You can add additional test points, if necessary, using the Tools|Test Points|Add command. To delete test points, either select the Edit|Undo command immediately after generating test points, or use the Edit|Delete command and set the selection filter to delete only flashes on the test_top or test_bot layer where the test points are placed.
14. Select the Tools|Test Points|Flying Probe|IPC-D-356A command to save your test file.
15. Specify the name of the .net file you are exporting. To search for a file or directory, click the Browse button.
16. Specify the Net To Net Adjacency Distance. Any nets within this distance to a net being tested will be tested for shorts.
17. Click Save to save the file.

Working with Embedded Passives

When importing Gerber or other "dumb" file formats, the only information that VisualCAM sees are draws and flashes on layers. The system can't determine on its own what is or isn't a component. In order for the system to correctly analyze or modify embedded passives in your database, you must "reverse engineer" your data. This tutorial explains the basic steps to reverse engineer embedded passives. For detailed explanations of the commands involved in this process, click on the links provided.

To reverse engineer embedded passive data

1. Import all of your embedded passive data.
2. Use the Setup|Layers command to organize and tag your layers.
3. Select one of the Tools|Convert|Drawn Pads commands and convert all drawn pads on your conductive layers to flashes. If this is not done, the system will not be able to properly build your embedded passives.
4. Select the Tools|Embedded Passives|Find command. Here, you provide information about your embedded passive and conductive layers, which the system uses to automatically "build" the embedded passives.

   Every flash that is tagged as a resistor on the passive layer is marked with an Ohms symbol.

   Resistor Flash

   Every embedded passive that has been built is highlighted with an outline.

   "Built" Embedded Passive

   A list of all your embedded passives appears in the Valid Embedded Passives list in the Navigator, by reference designator. You can view information about each embedded passive, and perform a variety of tasks. See the *Embedded Passives* topic for details.

   If you have flashes that are marked as resistors, but do not belong to a valid embedded passive, the termination bars fell outside your defined tolerance. They are listed in the Invalid Embedded Passives list in the Navigator, and you will need to manually construct the embedded passive using either the Construct Embedded Passive command in the Navigator's right-click menu, or the Tools|Embedded Passives|Build Embedded Passive command.
5. Select the Analysis|DRC/MRC command, and run the Embedded Passive checks on your data. This ensures that your data are properly aligned, sized, etc.
6. If you are using etched/subtractive technology to create your embedded passives, use the Tools|Embedded Passives|Etch Compensation command to factor in any necessary manufacturing tolerances.
7. You may now export your data in your desired file format. If you use the File|Save command to save the data as a VisualCAM .vcam database file, all of your embedded passive "intelligence" will be saved as well.

Creating Stencils

This tutorial explains how to enhance the paste stencil openings over the pads for improved solder paste application.

To create stencils

1. Open or import the design with the paste mask layers that you wish to create the stencils from.
2. Use the Tools|Paste Mask|Optimize command to take care of properly undersizing the pads.

   -or-

   If your design does not have paste mask layers, use the Tools|Paste Mask|Generate command to create them.
3. Select the Tools|Stencils|Setup Shapes command, and define the stencil shapes you want to use.
4. Select the Tools|Stencils|Manual Conversion command to teach the system which stencil shapes you want to associate with which original openings over pad pairs. After selecting each pair of openings, when you are asked whether you want to convert the matching openings, click No. It will still save the assignments you make, but the openings will not be converted yet.
5. Select the Tools|Stencils|Automatic Conversion command to specify which paste mask layers you wish to convert, what destination layers to place the stencil data on, and convert all the openings.

Tip: Once you have completed steps 3 and 4, you can save the stencil definitions and associations to a file by clicking the Save button in the Automatic Stencil Conversion dialog box. The next time you want to enhance a stencil using the same stencil shape definitions and associations, you can skip steps 3 and 4 above. Just select the Tools|Stencils|Automatic Conversion command, click the Load button to load the file, and run the conversion.

Reverse Engineering for Assembly

The following procedures explain how to reverse engineer data, to add part centroid locations, pin locations, part rotation, etc. to a database, for assembly.

⚠ *These functions cannot be used to add intelligence to Gerber data for the purposes of exporting to a CAD system database.*

There are several ways to reverse engineer your data for assembly:

- Using IPC-D-356 Data
- Using a Centroid File
- Using a Part Footprint Library
- Using a Silkscreen
- Manually

Reverse Engineering Using IPC-D-356 Data

Reverse engineering from IPC-D-356 data is the most automatic and reliable method. However, note that your IPC-D-356 file must contain reference designators and pin numbers.

1. Import or open the database you wish to reverse engineer. Note that because this is for assembly, you only need to load the top and/or bottom side layers, and through-hole drill layer (inner layers are not needed unless you are also reverse engineering for assembly test).
2. All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, select Tools|Convert|Drawn Pads to convert them to flashes.
3. Select the File|Import|Netlist|IPC-D-356 command and import the netlist that goes with your data.
4. Use Edit|Align Layers and make sure your netlist layer is aligned to your layer data.
5. Select the Assembly|Parts|Identify Using|IPC-D-356 Data command. You are warned if you have not compared your netlist to your data, to check for any discrepancies. Although running a netlist comparison is recommended, it is not necessary to complete the reverse engineering process.

   The system scans the database, constructs the Footprint Library, and identifies the part instances in the workspace. Any errors logged in the Analysis area of the Navigator, and may ultimately be put in a .txt report file. When the process is finished, you are given the option of viewing the report file.

⚠ *Tip: Pin connectors may be identified in the IPC-D-356 file for the purposes of the netlist, but they will show up as errors during reverse engineering because they are not components. These errors should be ignored.*

6. Part data, such as pin numbers and centroids, are displayed as "virtual entities". Use the View|Parts command to display/hide the part data, and the Assembly|Parts|Query command to view information on the parts in the Item Properties Display.
7. If any of the footprint data, such as pin number assignments, needs to be corrected, use the Setup|Footprint Library command to view the footprint information and make the changes. The Footprint Library can be used for any global changes that need to be made to all parts that share a particular footprint. If you have a footprint that is incorrectly assigned to an individual part, use the Assembly section of the Navigator to change the assignment. You can also use the Navigator to alter things like rotation, reference designators, or delete a part altogether.
8. If you need to identify any additional parts, see Manual Reverse Engineering.
9. Identify any fiducials using the Assembly|Fiducials|Identify command.
10. If you have a bill of materials (BOM) for this data, select the Assembly|Import BOM command to add any additional part or device information to the database prior to exporting. Note that if the bill of materials contains device names or part numbers that do not match those in the footprint library, they will be added as necessary.
11. If you are going to export your data in the FATF format, and will use it for in-circuit test, select the Tools|Netlist|Set Netlist Information command to specify whether your nets are power or signal. When you are finished, use the Assembly|Export FATF command to export your data in the FATF format, or the Assembly|Parts|BOM Report command to generate an ASCII report that includes BOM and Centroid information. You can also export your data using the File|Export|ODB++, File|Export|ODB++(X), File|Export|GenCAD, or File|Export|IPC-D-356 command, and the exported data will include component information.

Reverse Engineering Using a Part Footprint Library

VisualCAM has the ability to save and load its own footprint library file. If you have a standard footprint library that you use for multiple boards, you can use it in reverse engineering.

1. Import or open the database you wish to reverse engineer. Note that because this is for assembly, you only need to load the top and/or bottom side layers, and through-hole drill layer (inner layers are not needed unless you are also reverse engineering for assembly test).
2. All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, select Tools|Convert|Drawn Pads to convert them to flashes.
3. Select the Assembly|Setup Footprint Library command. The Footprint Library dialog box appears.
4. Click the Load button, and select the appropriate .fpl file to import.
5. Click the OK button in the Footprint Library dialog box.
6. Select the Assembly|Parts|Identify Using|Footprint Library command, and identify the part instances for each footprint.
7. If you have a footprint that is incorrectly assigned to an individual part, use the Assembly section of the Navigator to change the assignment. You can also use the Navigator to alter things like rotation, reference designators, or delete a part altogether.
8. If you need to identify any additional footprints and assign them to parts, see Manual Reverse Engineering.
9. Identify any fiducials using the Assembly|Fiducials|Identify command.
10. If you have a bill of materials (BOM) for this data, select the Assembly|Import BOM command to add any additional part or device information to the database prior to exporting.
11. If you are going to export your data in the FATF format, and will use it for in-circuit test, select the Tools|Netlist|Set Netlist Information command to specify whether your nets are power or signal.
12. When you are finished, use the Assembly|Export FATF command to export your data in the FATF format, or the Assembly|Parts|BOM Report command to generate an ASCII report that includes BOM and Centroid information. You can also export your data using the File|Export|ODB++, File|Export|ODB++(X), File|Export|GenCAD, or File|Export|IPC-D-356 command, and the exported data will include component information.

Reverse Engineering Using a Centroid File

Part Centroid files can be used to reverse engineer part data for assembly. A converter file is also necessary, which is used to instruct the system how to read the data in the various columns of the centroid file. If you do not have a converter file, you can easily create a new one using the interactive Centroid File Converter.

1. Import or open the database you wish to reverse engineer. Note that because this is for assembly, you only need to load the top and/or bottom side layers, and through-hole drill layer (inner layers are not needed unless you are also reverse engineering for assembly test).
2. All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, select Tools|Convert|Drawn Pads to convert them to flashes.
3. Select the Assembly|Parts|Identify Using|Centroid File command, identify the centroid file you wish to use, and complete the part identification process.
4. Part data, such as pin numbers and centroids, are displayed as "virtual entities". Use the View|Parts command to display/hide the part data, and the Assembly|Parts|Query command to view information on the parts in the Item Properties Display.
5. If any of the footprint data, such as pin number assignments, needs to be corrected, use the Assembly|Setup Footprint Library command to view the footprint information and make the changes. The Footprint Library can be used for any global changes that need to be made to all parts that share a particular footprint. If you have a footprint that is incorrectly assigned to an individual part, use the Assembly section of the Navigator to change the assignment. You can also use the Navigator to alter things like rotation, reference designators, or delete a part altogether.
6. If you need to identify any additional parts, see Manual Reverse Engineering.
7. Identify any fiducials using the Assembly|Fiducials|Identify command.
8. If you have a bill of materials (BOM) for this data, select the Assembly|Import BOM command to add any additional part or device information to the database prior to exporting.
9. If you are going to export your data in the FATF format, and will use it for in-circuit test, select the Tools|Netlist|Set Netlist Information command to specify whether your nets are power or signal.
10. When you are finished, use the Assembly|Export FATF command to export your data in the FATF format, or the Assembly|Parts|BOM Report command to generate an ASCII report that includes BOM and Centroid information. You can also export your data using the File|Export|ODB++, File|Export|ODB++(X), File|Export|GenCAD, or File|Export|IPC-D-356 command, and the exported data will include component information.

Reverse Engineering Using a Silk Screen

You can use silkscreen layers to reverse engineer your part data for assembly. This function requires that the reference designators in your silkscreen be "real text", so you will have to do some processing of your silkscreen layer.

1. Import or open the database you wish to reverse engineer. Note that because this is for assembly, you only need to load the top and/or bottom side layers, their associated silkscreen layers, and the through-hole drill layer (inner layers are not needed unless you are also reverse engineering for assembly test).
2. All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, select Tools|Convert|Drawn Pads to convert them to flashes.
3. Select the Tools|Convert|Drawn Text command on your silkscreen layers to convert your reference designators to "real text".
4. Select the Assembly|Parts|Identify Using|SilkScreen Layer command, and identify the parts.
5. If any of the footprint data, such as pin number assignments, needs to be corrected, use the Assembly|Setup Footprint Library command to view the footprint information and make the changes. The Footprint Library can be used for any global changes that need to be made to all parts that share a particular footprint. If you have a footprint that is incorrectly assigned to an individual part, use the Assembly section of the Navigator to change the assignment. You can also use the Navigator to alter things like rotation, reference designators, or delete a part altogether.
6. If you need to identify any additional parts, see Manual Reverse Engineering.
7. Identify any fiducials using the Assembly|Fiducials|Identify command.
8. If you have a bill of materials (BOM) for this data, select the Assembly|Import BOM command to add any additional part or device information to the database prior to exporting.
9. If you are going to export your data in the FATF format, and will use it for in-circuit test, select the Tools|Netlist|Set Netlist Information command to specify whether your nets are power or signal.
10. When you are finished, use the Assembly|Export FATF command to export your data in the FATF format, or the Assembly|Parts|BOM Report command to generate an ASCII report that includes BOM and Centroid information. You can also export your data using the File|Export|ODB++, File|Export|ODB++(X), File|Export|GenCAD, or File|Export|IPC-D-356 command, and the exported data will include component information.

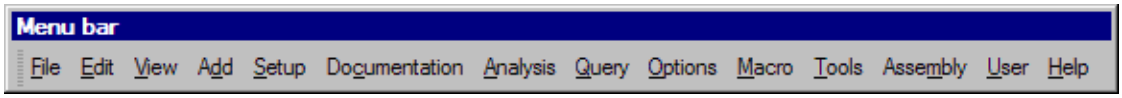
Manual Reverse Engineering

If you do not have an IPC-D-356 netlist, centroid file, footprint library, or silkscreen layers for your database, you can manually identify parts using this function. For the sake of speed and accuracy, if you do have any one of the aforementioned data files, it is recommended that you use it to reverse engineer your part data for assembly.

This function can also be used after you have used any of the other Part Identification commands, and had to correct any data errors that required deleting a part instance and recreating it.

1. Import or open the database you wish to reverse engineer. Note that because this is for assembly, you only need to load the top and/or bottom side layers, and through-hole drill layer (inner layers are not needed unless you are also reverse engineering for assembly test).
2. All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, select Tools|Convert|Drawn Pads to convert them to flashes.
3. Select the Assembly|Parts|Identify Using|Select Footprint command, and identify the parts.
4. If any of the footprint data, such as pin number assignments, needs to be corrected, use the Assembly|Setup Footprint Library command to view the footprint information and make the changes. The Footprint Library can be used for any global changes that need to be made to all parts that share a particular footprint. If you have a footprint that is incorrectly assigned to an individual part, use the Assembly section of the Navigator to change the assignment. You can also use the Navigator to alter things like rotation, reference designators, or delete a part altogether.
5. Identify any fiducials using the Assembly|Fiducials|Identify command.
6. If you have a bill of materials (BOM) for this data, select the Assembly|Import BOM command to add any additional part or device information to the database prior to exporting.
7. If you are going to export your data in the FATF format, and will use it for in-circuit test, select the Tools|Netlist|Set Netlist Information command to specify whether your nets are power or signal.
8. When you are finished, use the Assembly|Export FATF command to export your data in the FATF format, or the Assembly|Parts|BOM Report command to generate an ASCII report that includes BOM and Centroid information. You can also export your data using the File|Export|ODB++, File|Export|ODB++(X), File|Export|GenCAD, or File|Export|IPC-D-356 command, and the exported data will include component information.

VisualCAM Command Reference



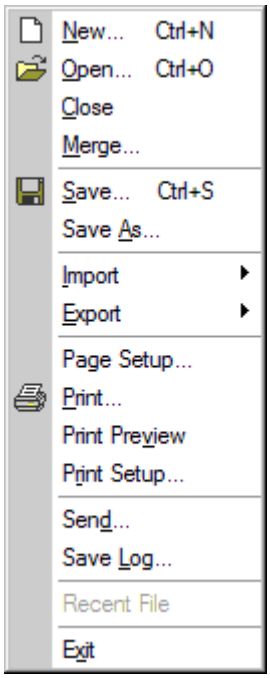
To view a menu, position your cursor on the Menu bar, and click on it with the left mouse button. This presents lists of commands that you can execute by clicking on them individually.

Terminating a Command
You may terminate a command, or at least one level of a multi-step command, by pressing the Esc key on your keyboard. You may also right-click and choose Cancel from the right-click shortcut menu. If you would like to terminate a command using the right mouse button, you can disable the right click shortcut menu. See the Options|Configure command.

Selecting another command from the Menu bar will also terminate any active command. Hotkeys, however, will not terminate an active command.

File Menu

The File menu is used for working with files and printing.



File|New

Toolbar Button: 

Equivalent Hotkey: Ctrl+N

The New command clears the current workspace and allows you to create or open a new design.

File|Open

Toolbar Button: 

Equivalent Hotkey: Ctrl+O

This command allows you to open an existing VisualCAM (.vcam) or GerbTool (.gtd) design database. Use one of the File|Import commands to open databases in other formats, such as Gerber.

This command can also be used to import CAM350 (.pcb or .cam file) databases. CAM350 files can also be imported with the File|Import|CAM350 command.

To open a VisualCAM database

1. Select the File|Open command. The Open dialog box appears.
2. Select the .vcam or .gtd file you wish to open.

   -or-

If you wish to open a CAM350 file, click on the Files Of Type box and select the CAM350 Database option. This will allow you to search for and select a .pcb or .cam file.

3. Click the Open button to open the database in the workspace.

Notes regarding CAM350 files:

- CAM350 v7.X databases can contain panelized data. If your database contains a panel, the panel itself, the one-up image, and any symbols are saved as separate .vcam files (you are prompted to select a folder to contain the files). You are then asked if you want to view the panel or the one-up image file.
- CAM350 DFM errors are added as Redline data (on a separate layer).
- Polygon Voids are supported, but are not recognized as standalone entities. Voids are only relative to the closed and filled polygon that contains them. If you query a polygon void (Query|Item command), the information returned is for the polygon itself. Polygon voids also cannot be modified, added, or deleted.

File|Close

This command closes the current design file. The Event Log can still be viewed after closing a file.

If you also wish to exit the VisualCAM application, use the File|Exit command instead.

To close a file

1. Select the File|Close command.
2. If you have any changes that have not been saved, you are asked whether you wish to save your changes to a .vcam file. If you do not save it, any changes you made since you last saved the file will be discarded. After you make your selection, the workspace is cleared and the Event Log is displayed.

After closing a file, if you wish to create a new database, you must select the File|New command. Otherwise, use File|Open or one of the Import commands, to open an existing database.

File|Merge

Selecting this command allows another complete VisualCAM (.vcam file), GerbTool (.gtd file) or CAM350 (.pcb or .cam file) to be merged layer by layer into the currently loaded design. The layers will be ordered exactly as they are in each design, so it is important that the layer types of each design match.

If you want the layers to remain separate, you must first open one of the designs and move the layers so that the two designs do not have any common layer numbers. Or, instead of using the Merge function, you can export one of the databases as Gerber (if is not already in that format), then use File|Import|Import Wizard to load the Gerber layers. Gerber files are always imported to empty layers.

To merge databases

1. Use File|Open or a File|Import command to load one design.
2. Select the File|Merge command. The Merge Design dialog box appears.
3. Select the .vcam or .gtd file you wish to merge in.

   -or-

   If you wish to merge in a CAM350 file, click on the Files Of Type box and select the CAM350 Database option. This will allow you to search for and select a .pcb or .cam file.
4. Click the Open button. The origin of the file is attached to the cursor, and you are prompted to select an insertion point in the workspace.
5. Click where you want the merged design to be.

   -or-

   Click the Abs button in the XY Bar, or press the C hotkey, and specify the exact coordinates where the design should be placed (e.g. 0:0).
6. You are warned of any errors or layer type conflicts, and your databases are merged.
7. Press the Esc key to exit the function.

Notes regarding CAM350 files:

- CAM350 v7.X databases can contain panelized data. If your database contains a panel, the panel itself, the one-up image, and any symbols are saved as separate .vcam files (you are prompted to select a folder to contain the files). You are then asked if you want to view the panel or the one-up image file.
- CAM350 DFM errors are added as Redline data (on a separate layer).
- Polygon Voids are supported, but are not recognized as standalone entities. Voids are only relative to the closed and filled polygon that contains them. If you query a polygon void (Query|Item command), the information returned is for the polygon itself. Polygon voids also cannot be modified, added, or deleted.

File|Save

Toolbar Button: 

Equivalent Hotkey: Ctrl+S

Select this command to save the current design as a VisualCAM database (.vcam) file. This command does not clear the current workspace; you may continue to work on the current design after saving. It does not overwrite any files that may have been imported.

Tip: VisualCAM .vcam files cannot be opened by GerbTool. If you wish to save your data so that it can be opened by GerbTool version 15.X, use the File|Export|GerbTool command.

Related topics

File|Export

File|Save As

Select this command to save the current design under a different filename. The design is saved in the VisualCAM .vcam database format. This command does not clear the current workspace; you may continue to work on the newly named design after saving. It does not overwrite any files that may have been imported.

Related topics

File|Export

File|Import

The Import commands are used to open database files of various formats. If you wish to open a VisualCAM .vcam file, use the File|Open command instead.

File|Import|Import Wizard

This command opens the Import Wizard. The Wizard allows you to specify which files (Gerber, aperture lists, VisualCAM databases, Drill files, etc.) you wish to import, assign aperture lists to specific Gerber files, and order the loading sequence. VisualCAM automatically recognizes the file types, and imports the specified files into the first empty block of consecutive layers found. D-codes with multiple definitions are automatically remapped to new D-codes, as needed.

To use the Import Wizard

1. Make sure all the files you wish to import are in the same folder.
2. Select the File|Import|Import Wizard command. Page 1 of the Import Wizard appears:

   My Data Is In
   Enter the name of the folder containing your files to import. Click the Browse button to search for the folder.

   Configure File Types To Ignore
   This button opens the Configuration dialog box, where you may enter any file name extensions that you want the program to ignore when scanning the folder. The more files that may be safely ignored, the faster the Import Wizard will be able to scan the specified folder.

   Configure Aperture List Converters
   This button opens the Configuration dialog box, where you may specify which aperture list converters should be enabled. The more aperture list converters enabled, the longer it will take to scan the specified folder.
3. Click the Next button to continue to Page 2 of the Import Wizard. This page presents the list of files that VisualCAM found in the selected import folder. Click on the list column headers to sort the list based on the data within the respective columns.

   Filename
   Indicate whether VisualCAM should import a file by putting a check mark by the filename. Click the Unselect All Importable button if you wish to remove the check marks next to all file names.

   File Type
   All importable file types are indicated (if a file cannot be imported, "???" appears.) If VisualCAM detects the file type incorrectly, you may change it by clicking on the file type in question, and selecting the correct type from the shortcut menu.

   Data Format
   This column indicates the detected data format for each file. To override the detected format, click on the appropriate entry.

   - If the file is Gerber, the Gerber Import Data Format dialog box appears. See File|Import|Gerber for details on using this dialog box.

Tip: In order for a 274-X file to be properly recognized, it must have an %FS statement (Format Statement) and at least one %ADD statement (Aperture Definition). If you know your files are 274-X but are not recognized as such, you should check your file in a text editor for these statements.
If the Format Statement is missing, you can either manually add it to the file in the text editor, or specify the settings in the Gerber Import Data Format dialog box. The file should then import without further issues.
If the Aperture Definition statement is missing, you can specify the format settings in the Gerber Import Data Format dialog box. The file will import without any aperture definitions, which you can add later (if desired) using the Setup|Apertures command.
If you need further information regarding Gerber file formats, see Gerber for Beginners*.*

- If the file is an aperture list, the Select Converter dialog box appears. VisualCAM has already selected an appropriate aperture converter to use for your aperture list, if one is available. To override the selection, select a different converter to use from the Data Format list. You can have your selection applied to all incoming aperture lists by clicking the Apply To All button. You can create a new converter by clicking the New Converter button. See File|Import|Aperture List for details on creating a converter.
- If the file is an NC tool list, the Select Converter dialog box appears. VisualCAM has already selected an appropriate converter to use for your tool list, if one is available. To override the selection, select a different converter to use from the Data Format list. You can have your selection applied to all incoming tool lists by clicking the Apply To All button. You can create a new converter by clicking the New Converter button. See File|Import|NC Tool List for details on creating a converter.
- If the file is DXF, the Import DXF dialog box appears. See File|Import|DXF for details on using this dialog box.
- If the file is HPGL, the HPGL Import dialog box appears. See File|Import|HPGL for details on using this dialog box.
- If the file is DPF, there are no format parameters to modify.
- If the file is NC (drill or mill) data, the Import NC Data dialog box appears. See File|Import|NC (Drill/Mill) for details on using this dialog box.

Length
The size of the file, in bytes.

Date
The date that the file was last modified.

Show Unknown File Types
By default, the Wizard only displays those files it can identify. Select this option if you wish to see all files contained in the selected import folder.

4. Click the Next button to continue.
5. If you are importing any 274-D files, Page 3 appears. This page is used to assign 274-D Gerber files to aperture lists. Drag each Gerber file onto the desired aperture list. (If only one aperture list is present, all Gerber files are automatically assigned to it.) Then click the Next button.

   If you are not importing any 274-D files, Page 3 is skipped.
6. Page 4 presents the final list of files that are to be imported. The files are imported to layers in the order they are listed. If you wish to reorder the files so that your layers are in the proper order, click and drag each file to the desired position.

   If you wish to prevent warning messages from being displayed when importing files, select Suppress Import Warning Messages. A log file will still be created but you will not be prompted whether to view it.
7. When you are satisfied with your selections and ordering, click the Finish button. Your files are imported and any pertinent information or warning messages are displayed in the Log screen. You can save the log by right-clicking on the Log screen, and selecting the Save Log command from the shortcut menu.

Tip: Polygon Voids in 274-X files are supported, but are not recognized as standalone entities. Voids are only relative to the closed and filled polygon that contains them. If you query a polygon void (Query|Item command), the information returned is for the polygon itself. Polygon voids also cannot be modified, added, or deleted.

Related topics

Using Files
ACR Files
Aperture List Files

File|Import|Gerber

This command allows you to import one or more Gerber files into the first empty block of consecutive layers available the current database. All files selected must be of the same file format. If you have files of a different format, you must repeat the use of this command for each type of file, or use the Import Wizard.

Tip: Polygon Voids in 274-X files are supported, but are not recognized as standalone entities. Voids are only relative to the closed and filled polygon that contain them. If you query a polygon void (Query|Item command), the information returned is for the polygon itself. Polygon voids also cannot be modified, added, or deleted.

To import Gerber files.

1. If the files being imported are in the 274-D format, Use the File|Import|Aperture List command to import the Aperture list that goes with your Gerber files. Gerber files in the extended formats (like 274-X) contain their own aperture information, so no aperture list needs to be imported.

   If you already have a database loaded, incoming D-codes with duplicate numbers are automatically remapped to new D-codes, as needed.

2. Select the File|Import|Gerber command. an Import dialog box appears.
3. Click the file you wish to import. If you wish to select more than one, press and hold down the Ctrl key and click on each file name. Then click the Open button to confirm your selection. The Gerber Import Data Format dialog box appears.
4. The system has already run a series of checks to determine the format of your files. The detected format is displayed in the Gerber Import Data Format dialog box. If you make any changes, click the Apply button to see those changes reflected in the thumbnail view of the file. This allows you to verify the accuracy prior to actually importing the files.

   Dialect
   Indicates the specific format of the Gerber file: 274-D, 274-X, or FIRE9XXX. In order for VisualCAM to identify a file as 274-X, the FS statement and at least one ADD statement must be present in the file. If either statement is missing, it will be identified as 274-D.

   If you think a file is 274-X, but is identified by VisualCAM as 274-D open the file in a text editor to verify the format.

   If the Format Statement is missing, you can either manually add it to the file in the text editor, or specify the settings in the Gerber Import Data Format dialog box. The file should then import without further issues.

   If the Aperture Definition statement is missing, you can specify the format settings in the Gerber Import Data Format dialog box. The file will import without any aperture definitions, which you can add later (if desired) using the Setup|Apertures command.

   If you need further information regarding Gerber file formats, see *Gerber for Beginners*.

   M.N
   This is the coordinate format. For example, a 2.3 format specifies two decimal digits before an implied decimal point and three following. (12250 represents 12.250 if the coordinate format is 2.3).

   Terminator
   Indicate the block terminator (EOB) as either * or $.

   Coordinate Mode
   Absolute mode is when all X:Y coordinates are referenced to a common origin. Incremental mode is when each X:Y coordinate is a displacement from the previous coordinate.

   Zero Suppression
   No zero suppression allows coordinates to have zeroes in front or back of integers. Leading zero suppression shows zeroes only at the end of a number, trailing zero suppression shows zeroes only at the beginning of a number.

   Character Set
   Indicate the character text used in the file.

   Special
   You may enable Metric mode indicating that your files are in metric format.

   If your files have Step & Repeat codes, you can choose Flatten S&R to import all the stepped and repeated images as "real data" (for example, you are importing a panel and want to have all the images editable, instead of "virtual").

Select Honor CR/LF if carriage return and line feeds should be honored as block terminators.

The Merge Polygons option controls how Polygons are handled (G36 & G37 commands, in Gerber). If you select Merge Polygons, files with multiple G36/G37 polygons that touch or overlap are combined: If positive data, 2 or more polygons become 1, if a positive polygon is overlapped by a negative polygon, it becomes a positive polygon with a void. If you do not select this option, all polygons will be individual entities. Dark and clear polarity are maintained, though the clear items will appear black in the workspace.

Assign 274-X Polarity To
If you are importing 274-X files, you can have polarity managed in two different ways. Individual Items imports each file as a single layer, and assigns polarity on an item-by-item basis. If Layer Composites is selected, then a layer with the correct polarity is created for every logical layer (LP command) in the imported file, resulting in a composite.

Arcs
Specify whether all circular interpolated arcs should be considered 360°, whether redundant coordinates should be removed from arcs, whether to convert small arcs to linear line segments, and the chord angle used in segmenting arcs found in filled polygons.

If small arcs are to be converted to linear line segments, the two parameters that affect this conversion are the angle and the delta. Min Arc Angle specifies the smallest sweep angle that an arc may be before it is selected for segmentation. Min Arc Delta specifies the smallest distance between end points of an arc before it is selected for segmentation.

Set Min/Max Board Size
VisualCAM attempts to determine the size of your design based upon various values in the file. The dimensions are indicated in the dialog box above the thumbnail view of the design. If the indicated size is significantly larger or smaller than the actual design, you can specify a size range for your design. VisualCAM will then recalculate the size, based upon those limits. If your boards typically fall within a certain size range and use the same data format, specifying a minimum and maximum board size value and saving the settings as the default values can aid the import process in the future.

Apply To All
Applies the selected settings to all imported files.

Apply To
This is the directory path and name of the Gerber file you are importing.

Set As Default
Saves the current settings as the default settings for Gerber files whose format cannot be identified by VisualCAM. If an imported file can be identified by VisualCAM, the dialog box settings are set accordingly (not to the default settings).

Use Default
Applies the saved default settings to the imported files, overriding the detected format. If you use standard settings, take note of the thumbnail view prior to completing the import process. If a design is loaded with an incorrect format specified, VisualCAM will attempt to display it with unpredictable results. If you inadvertently import a Gerber file incorrectly, use the Undo command and re-import.

5. When you are finished, click the OK button to import your files.

Related topics

Using Files
ACR Files
Aperture List Files

File|Import|Aperture List

274-D Gerber data requires a separate aperture list be imported before the design files. This command allows you to import an aperture list file.

If you import an aperture list into a design that already has apertures defined, all D-codes that were previously defined will be remapped to new D-codes and an internal remapping table will be created. Therefore, any Gerber files subsequently imported with the Import|Gerber command will have their D-codes remapped accordingly. This procedure of importing an aperture list followed by one or more basic Gerber files may be repeated as many times as needed, as long as there is room for more apertures. The Import Wizard makes easy work of importing multiple sets of Gerber files, each with their own aperture list, in one simple session.

Tip: If you are importing a VisualCAM Aperture List (.map file), any custom apertures are imported as moirés. Use the Custom Ap button in the Aperture Setup dialog box to load or define your custom apertures. Use of 274-X, or saving your data as a VisualCAM database (.vcam file), will avoid this issue.

To import an aperture list

1. Select the File|Import|Aperture List command. The Import Aperture List dialog box appears.
2. In the Filename box, specify the file to import. Click the browse button if you need to search for the desired file.
3. Choose the aperture list Automatic Conversion Rule (ACR file) to use from the Format list. If you are not sure which format to choose, select the Auto Detect option to have the format determined automatically.

   All ACR files that are provided with VisualCAM are in the Apconv folder. If you have an ACR file that you would like to appear in the Format list, see Options|Configure for information.

   If no ACR file exists, you can either create your own using a text editor, or create one interactively in VisualCAM. You can also Edit an existing file, by clicking the Edit Converter button. See *Creating an Aperture List Converter*.
4. When you are finished making your selections, click the OK button. Your apertures are imported/converted, and are added to the Aperture Table, where they can be viewed and edited.

Creating an Aperture List Converter

VisualCAM provides an interactive ACR File Creator that allows you to quickly create an aperture converter file.

1. In the Import Aperture List dialog box, click the New Converter button. The Create Aperture List Converter dialog box appears. The name of the aperture list being imported appears at the top, and your current aperture list is displayed in the Format 1 tab.
2. Your aperture list appears in a spreadsheet, and default column names are provided at the top. These columns are not automatically matched to the aperture list; you must reorder and resize the columns to match the aperture list contents.
   - To move a column header, click it with your mouse and drag it to the correct location.
   - The column header width should exactly match the width of the aperture list columns. Click on the left or right edge of the column header and drag it to the appropriate location.
   - If you wish to delete a column header, right-click on it and select Delete Column from the shortcut menu. Note that once you delete a column, it can always be added again later (it appears as an Add command in the right-click menu).
   - If you have a column of information in your aperture list that you wish to be ignored, right-click on the column header currently above it, and select Add Skip Column from the shortcut menu. A header titled "Skip" now appears above the contents.

   If you find that you have an aperture list that contains mixed definitions, where not all aperture types use the same column format, see the Mixed Format Definitions section, below, for details on how to manage these types of lists.
3. Specify the Delimiters used to separate the columns in your aperture list.
4. Select the Units of measure for your aperture list.
5. If you have any lines in the aperture file, such as headers comments, specify in the Initial Lines To Ignore box how many lines at the beginning of the file should be ignored by the aperture converter. Also specify, in the Comment Text box, the symbol that leads any comment lines that may be intermixed with your aperture information.
6. Right-click on the shape names in your aperture list, and select the corresponding shape from the right-click menu. (Note that "Therm45" is a Thermal whose spokes are rotated 45 degrees.) VisualCAM automatically recognizes all D-codes with the exact same shape name. The recognized aperture definitions are then listed in the Matched Apertures list. All recognized D-codes are also colored red in the Aperture File view. Continue defining the shapes until they are all listed.

⚠ *Any apertures that are not matched will not be defined. After importing your 274-D files, you will need to use the Setup|Apertures command to define those apertures.*

7. When you are finished, click the Save Converter button to save your ACR file in the Apconv folder. The dialog box closes and your converter appears in the Format list of the Import Aperture List dialog box.

Mixed Format Definitions

An aperture list that contains mixed definitions, such as the one below that has only an X size for D-code 10, and X and Y size for D-code 11, and X, Y, and rotation value for D-code 12.

```
D10   ROUND   .010   INCH
D11   RECT    .010   .020   INCH
D12   THERM   .090   .060   45     INCH
```

If you have a similar case, you can create multiple Format types to handle the information.

1. Create the column definitions for the first type of format (e.g. D-code 10's format), as instructed above in steps 1-4.

2. Right-click on the Shape type (e.g. ROUND), and select the matching shape definition from the right-click menu. VisualCAM immediately recognizes all other D-codes with identical shapes, turns the text for those D-codes red, and lists them in the Matched Apertures list. Continue matching all shapes until you are finished.
3. Right-click on the Format 1 tab, and select Add Format from the shortcut menu. A Format 2 tab appears. The D-codes that have already been recognized based on the Format 1 definitions are listed in gray text.
4. Modify the column headers so that they are appropriate for the second type of formatting (e.g. D-code 11's format) and repeat the same steps as you did for Format 1. You can create as many Format types as necessary to convert your entire aperture list.
5. When you are finished, click the Save Converter button to save your ACR file in the Apconv folder. The dialog box closes and your converter appears in the Format list of the Import Aperture List dialog box.

Related topics

Gerber for Beginners
Working with Apertures
Aperture List Files

File|Import|NC (Drill/Mill)

This command allows you to import an NC (drill or mill) file to a layer that you specify. The format of the file will be automatically detected for you, but you are given an opportunity to review and change any of the format settings prior to loading the file.

Importing PADS Drill Files
PADS drill files do not include the drill tool definitions. Instead, a separate .rep file, containing the definitions, is created. If you are importing a drill file from a PADS CAD system, complete the following steps before you import the file.

1. Make sure that the following PADS ACR files are currently in the VisualCAM "ToolConv" folder:

   PadsMil.acr
   PadsInch.acr
   PadsDrillMM.acr

2. Select the File|Import|NC Tool List command and import the .rep file that corresponds with your PADS drill file.

To import an NC file

1. Select the File|Import|NC (Drill/Mill) command. The Import NC dialog box appears.

Tip: If your file name has an extension that does not appear in the "Files of type" list, select the Options|Configure command, and click on the Paths, Files, Extensions tab. Add your file extension to the Default Extensions list for your Drill or Mill files.

2. Select the file to import, and click the Open button. The Import NC Data dialog box appears.
3. In the Destination Layer box, select the layer to place the NC data.
4. Select an NC Tool Table to place your tool definitions. You can create a new tool table, or add to an existing tool table (if another database is already loaded).
5. Because NC files do not contain decimal points, a number such as X19485 could mean 1.9485 inches, 19.485 inches, or 194.85 inches. To help limit the possibilities, the system assumes that the minimum board size is 2.5 inches and the maximum size is 25 inches. With these base values, it can then determine the accurate coordinate (M.N) format and size of your board.

   The detected board dimensions are indicated above the thumbnail view of the design. If the indicated size is significantly larger or smaller than the actual design, you can click the Set Min/Max Board Size button and specify a more accurate range for the board. VisualCAM will then recalculate the size, based upon those limits. If your boards typically fall within a certain size range and use the same data format, specifying a minimum and maximum board size value and saving the settings as the default values can aid the import process in the future.
6. Because the file format is automatically detected, it should not be necessary to change any of the other file format settings. However, if you need to make any changes, you should click the Apply button to see those changes reflected in the thumbnail view of the file. This allows you to verify the accuracy prior to actually importing the file into your database.

   Dialect
   The format of the file to be imported. Available options are Excellon, Excellon 2, Sieb & Meyer 1000, Sieb & Meyer 3000, and Takeuchi

M.N
The coordinate format. For example, a 2.3 format specifies two decimal digits before an implied decimal point and three following. (12250 represents 12.250 if the coordinate format is 2.3).

Mode
Absolute mode is when all X:Y coordinates are referenced to a common origin. Incremental mode is when each X:Y coordinate is a displacement from the previous coordinate.

Zero Suppression
No zero suppression allows coordinates to have zeroes in front or back of integers. Leading zero suppression shows zeroes only at the end of a number. Trailing zero suppression shows zeroes only at the beginning of a number.

Units
Select if the units of measure are in Inches or Metric.

Character Set
Indicate the character text used in the file.

Plating
Select Plated if the NC data on the layer are plated, Nonplated if the NC data are unplated, or Both if you have a combination of plated and unplated data on the layer. Your tools will be defined accordingly, in the tool table.

Step and Repeat
When stepping and repeating a pattern, Excellon supports repeated placement of the pattern with a fixed offset by using the repeat ("Rn") command in conjunction with the pattern offset command ("M02"). This option is provided because of discrepancies between the Excellon documentation and the behavior of some machine controllers. Selecting this option specifies that the result of "n" in the command "RnM02" would be the total number instances of the pattern (per Excellon documentation). So the result of "R5M02" would be 5 instances of the pattern. If not selected, the command will produce n+1 instances of the pattern; "R5M02" would result in 6 instances of the pattern (the original plus 5).

Apply To
This is the directory path and name of the file you are importing.

Format Sample
This displays the text version of your file. You cannot edit the contents of the file here—this is only for your reference.

7. If you wish to save the current settings as the defaults for NC files whose format cannot be identified by VisualCAM, click the Set As Default button. If an imported file can be identified, the format settings will be set accordingly (not to the default settings that you save). To apply the saved default settings to imported files, just click the Use Default button and any detected format settings are overridden.
8. Take note of the thumbnail view prior to completing the import process. If everything looks correct, click the OK button to import the file.

If data are imported with an incorrect format specified, they will be loaded with unpredictable results. If you inadvertently import a file incorrectly, use the Undo command and re-import.

If you import your data to an existing NC layer whose tool table already contains tool definitions, you will be prompted to resolve any conflicts. You are also warned if any items have no associated tool defined. Select the Setup|NC Tools command to view which tools are currently in use, and to add or change tool definitions.

File|Import|NC Tool List

This command allows you to import a Drill Tool List file. Tool list conversion is accomplished using the same ACR rule file technology as used by the Import|Aperture List command. If you already have a tool list converter you wish to use, copy the associated conversion rule file into the "ToolConv" folder (located in the main VisualCAM installation folder).

Tip: This is only used for importing tool lists that were created by other programs. If you are wish to load a VisualCAM .nct file, use the Setup|NC Tools command, and click the Load button in the NC Tool Setup dialog box.

To import an NC tool list

1. Select the File|Import|NC Tool List command. The Import NC Tools dialog box appears.
2. Click the Browse button to search for the tool list file you wish to import.
3. Select the tool list format from the Format list. If no converter file exists, you can either create your own using a text editor, or VisualCAM to create one interactively. See *Creating a Tool List Converter*.
4. You can either create a new tool table, or have your tool definitions added to an existing tool table (if you already have a database loaded). Make your selection from the Destination list.
5. Click the OK button to import the tool list. You can view or edit your tool definitions in the NC Tool Table.

Creating a Tool List Converter

VisualCAM provides an interactive Conversion File Creator that allows you to quickly create a tool list converter file.

1. In the Import NC Tools dialog box, click the New Converter button. The Create Tool List Converter dialog box appears. The name of the tool list file being imported is at the top, and your current tool definitions are displayed in the Format 1 tab.
2. Your tool list appears in a spreadsheet, and default column names are provided at the top. These columns are not automatically matched to the tool list; you must reorder and resize the columns to match the tool list contents.
   - To move a column header, click it with your mouse and drag it to the correct location.
   - The column header width should exactly match the width of the tool list columns. Click on the left or right edge of the column header and drag it to the appropriate location.
   - By default, only columns for the Tool Number and Size are provided. If your tool list contains additional parameters, such as Feed Rate or Depth Offset, you can add those columns by right-clicking in the appropriate location in the column heading, and selecting the appropriate Add command.
   - If you wish to delete a column header, right-click on it and select Delete Column from the shortcut menu. Note that once you delete a column, it can always be added again later.
   - If you have a column of information in your tool list that you wish to be ignored, right-click on the column header currently above it, and select Add Skip Column from the shortcut menu. A header titled "Skip" now appears above the contents.

   If you find that you have a tool list that contains mixed definitions, where not all tools use the same column format, see the Mixed Format Definitions section, below, for details on how to manage these types of lists.
3. Specify the Delimiters used to separate the columns in your tool list.
4. Select the Units of measure for your tool list.
5. If you have any lines in the tool file, such as headers or comments, specify in the Initial Lines To Ignore box how many lines at the beginning of the file should be ignored by the tool converter. Also specify in the Comment Text box the symbol that leads any comment lines in your tool list
6. Once the Tool Number and Size columns have been properly defined, the system automatically recognizes the information in the list and confirms it in the Matched Tools list. All recognized tools are also colored red in the Format list. When you are finished, click the Save Converter button to save your converter file in the ToolConv folder.
7. The dialog box closes and your converter appears in the Format list of the Import Tool List dialog box.

Mixed Format Definitions

A tool list that contains mixed definitions, such as the one below that indicates when hole is plated, but not when it is unplated.

```
;PLATED          Holesize 1. =   12.000000   MILS
;Holesize 2. =   79.000000       MILS
;PLATED          Holesize 3. =   23.000000   MILS
```

If you have a similar case, you can create multiple Format types to handle the information.

1. Create the column definitions for the first type of format (e.g. Tool 1 and 3's format), as instructed above in steps 1-4.
2. Once the Tool Number and Size columns have been properly defined, the system automatically recognizes the information in the list and confirms it in the Matched Tools list. All recognized tools are also colored red in the Format list.
3. Right-click on the Format 1 tab, and select Add Format from the shortcut menu. A Format 2 tab appears. The tools that have already been recognized based on the Format 1 definitions are listed in gray text.
4. Modify the column headers so that they are appropriate for the second type of formatting (e.g. tool 2's format) and repeat the same steps as you did for Format 1. You can create as many Format types as necessary to convert your entire tool list.
5. When you are finished, click the Save Converter button to save your converter file in the ToolConv folder. The dialog box closes and your converter appears in the Format list of the Import Tool List dialog box.

File|Import|Barco DPF

This command allows you to import one or more Barco DPF files.

To import a DPF file

1. If you want to import your file into a specific, empty layer, make that layer the active layer. If the active layer is not empty, a new layer following the active layer will be created for you and used. (As many layers as necessary will be created to import all of the files you specify.)
2. Select the File|Import|Barco DPF command. An Import dialog box appears, where you can select one or more .dpf files to import.
3. To select one file, click on it. To select more than one file, hold down the Ctrl key while you click on each one.
4. Click the Open button to import the files.

File|Import|HPGL

This command allows you to import an HPGL or HPGL/2 plot file into the currently active layer.

To import an HPGL file

1. Make the layer you wish to import the file to the active layer.
2. Select the File|Import|HPGL command. The Import HPGL dialog box appears.
3. Select the HPGL file you wish to import, and click the Open button. The HPGL Import dialog box appears.
4. The size of each pen, as defined in the HPGL file, appears. You can change the size, if desired.
5. The Plot Size settings are used to tell VisualCAM the type of plotter the plot file was intended for. The values actually refer to the location of the origin with respect to the plot. On an A/B plotter, the origin is in the lower left hand corner of the page, while in the C/D/E it is in the center of the page. Changing this setting will only effect the placement of the merged information, any data that will extend beyond the page will not be clipped.
6. To rotate the data 90 degrees counter-clockwise, select the Rotate 90 option.
7. Select the HPGL/2 Scaling option if you are importing an HPGL/2 file. The main difference between the HPGL and HPGL/2 formats is that the HPGL/2 format supports scaling.
8. Click the OK button to import the data.

File|Import|DXF

This command allows you to import a DXF file into your design. VisualCAM supports up to AutoCAD 2000.

Each layer contained within the DXF file can be mapped to one or more VisualCAM layers. This flexibility allows for duplicating information onto multiple layers when, for instance, a pad master layer is used which needs to be merged onto each layer containing traces. Likewise, more than one DXF layer may be mapped to a single VisualCAM layer. Layers may also be mapped by color so that items of the same color are merged together onto a single VisualCAM layer. This feature can be useful for viewing DXF files containing many colors or items which do not share the same color as the DXF layer in which they appear.

To import a DXF file

1. If you plan to use the "Map by Color" option, make sure the Max Layers setting in the General Configuration Options is set high enough to allow for the number of colors expected; otherwise all items of the colors that exceed the number of layers this will appear on the last (highest-numbered) VisualCAM layer. Many drawings use only colors in the range of 1 through 9; however, valid colors can be of any value in the range of 1 through 255.
2. Select the File|Import|DXF command. The Import DXF dialog box appears.
3. Click the Browse button next to the Input File box to select the DXF file to import.
4. The import function produces a report file, which contains a summary of the DXF file. Enter a name for the .txt file in the Report File box. By default, the report is saved in the Samples folder. Click the browse button to select a different folder.
5. Click the Browse button next to the Font Directory box to specify the directory in which SHX font and shape files are to be found. Standard SHX font files and SHX Unifont files are supported both for text and shape entities. If text within the DXF file refers to a font that is not present on your system, or the font file is of an unrecognized type, a standard font will be used in its place.
6. The Source DXF Layers list shows the layers within the DXF file that you can map to zero or more Destination Layers in VisualCAM. To map the layers, click and drag the source layers onto the desired Destination Layers. Multiple DXF layers can map to a single VisualCAM layer. Also, a single DXF layer can map to multiple VisualCAM layers.

   -or-

   Click the Map All To Current button to automatically map all DXF layers to the currently active layer. This is required if no layer table exists in the DXF file.

-or-

Click the Map Sequentially button to display a dialog box that allows you to sequentially map DXF layers to VisualCAM layers. You first specify the initial destination layer, and are given the option to exclude DXF layer 0 from the mapping.

-or-

Click the Map By Color option to have DXF file items mapped onto VisualCAM layers based on color. Items of color 1 (red) will appear on VisualCAM layer 1, those of color 2 (yellow) appear on VisualCAM layer 2, and so on.

The Clear Map button can be used if you want to remove all the mapping currently defined, and start over.

7. Specify the scale factor used during merging. To keep your data the same size as is set in your DXF file, keep the default scale factor of "1".
8. Specify the Line Width, in inches, to use for zero-width lines. The default width is 0.01 inches. Zero-width, closed polylines create filled polygons in VisualCAM.
9. Select the Place At Origin option if you wish to have the lower-left corner of the DXF design placed at the origin.
10. Select the Clear Merge Layer if you wish to empty all destination layers prior to receiving DXF information.
11. If you select Create Apertures, apertures for drawing lines are created when an equivalent aperture does not already exist in the currently loaded database (if there is one). The size of the new aperture is obtained from the DXF feature data. If this option is not selected, the next smaller existing aperture is used. If a next smaller aperture does not exist, then the smallest is used. If you do not select this option, no apertures already exist, and no block mapping is performed (see step 16), then all data are imported as zero-width lines, and assigned a round aperture of the size specified in the Line Width box.
12. Select Fill Polygons to convert DXF polygons to raster-filled polygons. Otherwise, the polygons will not be filled.

⚠ *Warning: If you plan to export your files in 274-D format, do not select the Fill Polygons option. Raster filling is not supported in 274-D.*

13. Select Metric to indicate that the coordinates in the DXF file are in millimeters. Do not select this option if coordinates are in inches.
14. For the Polylines To Draws options, select All to have your polylines converted to draws; select None to convert them to raster-filled polygons; or select If Width Less Than and specify a width limit (polylines with smaller widths are converted to draws, larger widths are converted to raster-filled polygons).
15. For the Circles Less Than option, circles smaller than the specified amount will be translated as round flashes. Circles larger than this amount will be treated as circular draws. Set the amount as desired.
16. The Map Blocks function is only used if two conditions apply: You already have another database loaded that contains aperture information, and the DXF file you are importing was originally produced by VisualCAM. If both of these conditions do not apply to you, just click the OK button, and your data are imported. You are finished.

    -or-

    If you have aperture information already loaded in VisualCAM, and your DXF file was originally produced by VisualCAM, click the Map Blocks button. The Map DXF Blocks dialog box appears, which allows you to map blocks in the DXF file to apertures in the currently loaded database.
17. When VisualCAM exports DXF data, it names its blocks in a manner that identifies the D-code number, shape, and size of each aperture. If you wish to map each aperture by D-code number to the existing apertures, click the Auto Map button. After the D-codes are mapped, if you click on a Source DXF Block, its corresponding Destination D-code will also be highlighted. If you wish to change any assignment, click on the Source DXF Block in question, then click on the Destination D-code you wish to map it to.

    -or-

    To manually map each block to its destination, first click on the desired Source DXF Block. Then click on its corresponding Destination D-code. After the D-codes are mapped, if you click on a Source DXF Block, its corresponding Destination D-code is also highlighted. If you wish to change any assignment, click on the Source DXF Block in question, then click on the Destination D-code you wish to map it to.

    If you wish to remove all mapping assignments at any time, click the Clear Map button.
18. Click the OK to return to the Import DXF dialog box.
19. Click OK to finish importing your DXF file.

File|Import|CAM350

This command allows you to open a CAM350 database. CAM350 files formats up to, and including, version 7.X are supported.

To import a CAM350 file

1. Select the File|Import|CAM350 command. The Import CAM350 dialog box appears.
2. Select the desired .pcb or .cam file, and click the Open button.

Notes regarding CAM350 files:

- CAM350 v7.X databases can contain panelized data. If your database contains a panel, the panel itself, the one-up image, and any symbols are saved as separate .vcam files (you are prompted to select a folder to contain the files). You are then asked if you want to view the panel or the one-up image file.
- CAM350 DFM errors are added as Redline data (on a separate layer).
- Polygon Voids are supported, but are not recognized as standalone entities. Voids are only relative to the closed and filled polygon that contains them. If you query a polygon void (Query|Item command), the information returned is for the polygon itself. Polygon voids also cannot be modified, added, or deleted.

Related topics

File|Open

File|Import|IPC-2581 (Offspring)

This command is used to import IPC-2581 (aka the Offspring format) files.

To import an IPC-2581 database

1. Select the File|Import|IPC-2581 (Offspring) command. The Import Offspring File dialog box appears.
2. Select the desired IPC-2581 .cvg file, and click the Open button. Your file is imported.

File|Import|ODB++

This command imports ODB++ database files. Reference Designators and Pin Numbers are assigned to User Data for each associated item. In addition, you have the option of automatically running a netlist comparison, to verify your design data against your original netlist.

Tip: Polygon Voids are supported, but are not recognized as standalone entities. Voids are only relative to the closed and filled polygon that contains them. If you query a polygon void (Query|Item command), the information returned is for the polygon itself. Polygon voids also cannot be modified, added, or deleted.

To import an ODB++ database

1. Select the File|Import|ODB++ command. A Browse for Folder dialog box appears.
2. Pick the Job Folder that contains all your ODB++ database files, and click OK. The Import ODB++ dialog box appears.
3. Select Yes to import the data as intelligent EDA data, or No to import the files as graphics only, without any intelligent design information.
4. If you select Yes, you can select the Assign Net Data To External Net Layer option. If you select this option, all netlist information will be copied onto a separate, External Net layer.
5. If you select Assign Net Data To External Net Layer, you have two additional options that allow you to automatically perform a netlist comparison: Generate New Netlist From Artwork creates a new Internal netlist from your artwork, and Run Netlist Compare automatically runs the Analysis|Netlist Compare function after you click OK.

   If you do not select the Generate New Netlist From Artwork option, the original "Internal" netlist information will remain so that, if you later change your data and wish to compare against your original netlist, you can generate a new netlist and select the Preserve Existing Net Names option to more easily identify where the differences are.

Tip: If you select to generate a new netlist, the system automatically creates the netlist without presenting you with any options, such as including single point nets. If you have special requirements for generating the netlist, do not select the Generate New Netlist From Artwork option, and instead run the Tools|Netlist|Generate function after you have imported your database.

6. Click OK, and your database is imported. If you selected the Assign Net Data To External Net Layer option, an External Net layer is created in addition to your other layers. Viewed by itself, this layer appears to contain no data. However, this layer contains the external netlist information, and links the external net points to the appropriate top/inner/bottom layer net test points. To view the points, turn on the associated top, bottom, and inner layers, and the external net layer. The net points are displayed in four ways for easy reference:


Top Layer Net Point


Bottom Layer Net Point


Inner Layer Net Point


Thru-hole Net Point

If you selected the Run Netlist Compare option, VisualCAM automatically displays discrepancies in the Navigator, which you may use to view the errors in the workspace and print reports. If you need to change any data to fix errors, you should then re-run the Netlist Compare command to verify that the fix was accurate. Once no more errors exist, you can use the Tools|Netlist|Apply External Net Names command to apply the external net names to the database, if you wish.

Related topics

Comparing Netlists Tutorial

File|Import|ODB++(X)

This command imports ODB++ database files that are in the XML format. Reference Designators and Pin Numbers are assigned to User Data for each associated item. In addition, you have the option of automatically running a netlist comparison, to verify your design data against your original netlist.

Tip: Polygon Voids are supported, but are not recognized as standalone entities. Voids are only relative to the closed and filled polygon that contains them. If you query a polygon void (Query|Item command), the information returned is for the polygon itself. Polygon voids also cannot be modified, added, or deleted.

To import an ODB++(X) database

1. Select the File|Import|ODB++(X) command. The Import ODB++(X) dialog box appears.
2. Click the browse button next to the Input Data File box select your ODB++ database file.
3. Under Import EDA Data, select Yes to import the data as intelligent EDA data, or No to import the files as graphics only, without any intelligent design information.
4. If you select Yes, you can select the Assign Net Data To External Net Layer option. If you select this option, all netlist information will be copied onto a separate, External Net layer.
5. If you select Assign Net Data To External Net Layer, you have two additional options that allow you to automatically perform a netlist comparison: Generate New Netlist From Artwork creates a new Internal netlist from your artwork, and Run Netlist Compare automatically runs the Analysis|Netlist Compare function after you click OK.

   If you do not select the Generate New Netlist From Artwork option, the original "Internal" netlist information will remain so that, if you later change your data and wish to compare against your original netlist, you can generate a new netlist and select the Preserve Existing Net Names option to more easily identify where the differences are.

Tip: If you select to generate a new netlist, the system automatically creates the netlist without presenting you with any options, such as including single point nets. If you have special requirements for generating the netlist, do not select the Generate New Netlist From Artwork option, and instead run the Tools|Netlist|Generate function after you have imported your database.

6. Click OK, and your database is imported. If you selected the Assign Net Data To External Net Layer option, an External Net layer is created in addition to your other layers. Viewed by itself, this layer appears to contain no data. However, this layer contains the external netlist information, and links the external net points to the appropriate top/inner/bottom layer net test points. To view the points, turn on the associated top, bottom, and inner layers, and the external net layer. The net points are displayed in four ways for easy reference:


Top Layer Net Point


Bottom Layer Net Point


Inner Layer Net Point


Thru-hole Net Point

If you selected the Run Netlist Compare option, VisualCAM automatically displays discrepancies in the Navigator, which you may use to view the errors in the workspace and print reports. If you need to change any data to fix errors, you should then re-run the Netlist Compare command to verify that the fix was accurate. Once no more errors exist, you can use the Tools|Netlist|Apply External Net Names command to apply the external net names to the database, if you wish.

Related topics

Comparing Netlists Tutorial

File|Import|PADS ASCII

This command imports PADS ASCII files from PowerPCB v5.0 or earlier. All database intelligence, such as layer names, layer structure, etc. is maintained. Reference Designators and Pin Numbers are assigned to User Data for each associated item. In addition, you have the option of automatically running a netlist comparison, to verify your artwork against your original netlist.

After importing the database, you can use VisualCAM to create and export an IPC-D-356/A netlist, export an ODB++ database, etc.

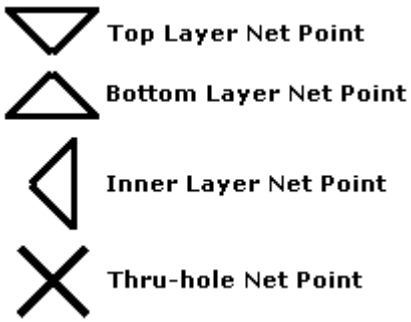
To import a PADS ASCII database

1. Select the File|Import|PADS ASCII command. The Import PADS ASCII File dialog box appears.
2. Select your desired .asc file and click Open. The Import PADS ASCII dialog box appears.
3. If you select the Assign Net Data To External Net Layer option, all netlist information will be imported onto a separate, External Net layer.
4. If you select Assign Net Data To External Net Layer, you have two additional options that allow you to automatically perform a netlist comparison: Generate New Netlist From Artwork creates a new Internal netlist from your artwork, and Run Netlist Compare automatically runs the Analysis|Netlist Compare function after you click OK.

   If you do not select the Generate New Netlist From Artwork option, the original "Internal" netlist information will remain so that, if you later change your data and wish to compare against your original netlist, you can generate a new netlist and select the Preserve Existing Net Names option to more easily identify where the differences are.

Tip: If you select to generate a new netlist, the system automatically creates the netlist without presenting you with any options, such as including single point nets. If you have special requirements for generating the netlist, do not select the Generate New Netlist From Artwork option, and instead run the Tools|Netlist|Generate function after you have imported your database.

5. Click OK, and your database is imported. If you selected the Assign Net Data To External Net Layer option, an External Net layer is created in addition to your other layers. Viewed by itself, this layer appears to contain no data. However, this layer contains the external netlist information, and links the external net points to the appropriate top/inner/bottom layer net test points. To view the points, turn on the associated top, bottom, and inner layers, and the external net layer. The net points are displayed in four ways for easy reference:



If you selected the Run Netlist Compare option, VisualCAM automatically displays discrepancies in the Navigator, which you may use to view the errors in the workspace and print reports. If you need to change any data to fix errors, you should then re-run the Netlist Compare command to verify that the fix was accurate. Once no more errors exist, you can use the Tools|Netlist|Apply External Net Names command to apply the external net names to the database, if you wish.

Related topics

Comparing Netlists Tutorial

File|Import|Netlist

The Import Netlist commands allow you to import an IPC-D-356/A netlist, or import only the netlist from an ODB++ or PADS ASCII database.

File|Import|Netlist|IPC-D-356

This command allows you to import and associate an IPC-D-356 and 356A format netlist into your design.

When importing IPC-D-356/A files, VisualCAM associates the net information for each layer in the input file to the conductive layers of the current design. Occasionally, you may encounter an IPC-D-356/A file that contains probe information for some of the inner layers. If you have a file of this type, you should make sure the appropriate layers are loaded in the current design. The layers will be associated in the same order they are encountered in the IPC-D-356/A file.

Since an IPC-D-356/A netlists contain information pertaining solely to pads and not traces, VisualCAM must generate an internal netlist prior to importing an IPC-D-356 netlist to ensure that your database contains a full and complete netlist after importing. While this may sound redundant, this prepares your data for an "automatic netlist comparison". This first step in the netlist comparison feature produces a report file detailing any differences between the internal and imported (external) netlist. When you run the Netlist Compare function, you are presented with a graphical comparison of the internal and external netlists, where VisualCAM highlights the discrepancies noted in the report. You will not have to "hunt and peck" through the report file and use the Query commands to find the issues.

If your goal is to perform a netlist comparison, see the *Comparing Netlists* tutorial for complete instructions. This topic will only cover the steps necessary for importing an IPC-D-356/A netlist.

To import a netlist

1. Use File|Import|Import Wizard to import your Gerber data or File|Open to a previously-created VisualCAM .vcam file.
2. If you opened a .vcam file that already contains a netlist, select the File|Import|Netlist|IPC-D-356 command to import the external netlist. Then continue with step 9.

   -or-

   If your database does not already contain a netlist, use the Setup|Layers command or the Navigator, make sure that each layer is "tagged" with the correct layer type. This is required so that the system understands what each layer is.
3. Remove any extraneous data on your electrical layers that are outside the perimeter of the board, such as title blocks, targets, crop marks, etc.

Tip: We recommend that any documentation for the board be done on a separate Drawing layer. If you wish to keep the data, placing it on a drawing layer assures it will not interfere with any processing, such as netlist extraction.

4. If you have any drawn pads in your database, select one of the Tools|Convert|Drawn Pads commands to convert them to flashes.
5. Use the Analysis|DRC/MRC command to make sure your layers are properly aligned. If any layers are misaligned, you can use the Edit|Align Layers command to align them.
6. Use the Tools|Pad Removal|Stacked command to remove any redundant pads.
7. Select the Tools|Netlist|Generate command, or use the Navigator shortcut command, to extract a netlist from your data.
8. Select the File|Import|Netlist|IPC-D-356 command, and select the desired netlist file from the Import IPC-D-356 dialog box.
9. After you import the netlist, a new External Net layer is created. Viewed by itself, this layer appears to contain no data. However, this layer contains the external netlist information, and links the external net points to the appropriate top/inner/bottom layer net test points. To view the points, turn on the associated top, bottom, and inner layers, and the external net layer. The net points are displayed in four ways for easy reference:


Top Layer Net Point


Bottom Layer Net Point


Inner Layer Net Point


Thru-hole Net Point

If net names are not provided in the IPC-D-356 file you imported, the system will automatically assign a unique net name as appropriate.

Error Messages

Below are examples of possible error message that can come from importing an IPC-D-356/A file, and what they mean:

No IPC data for location 2.8750, 3.7500 Layer:1

There is a pad on this layer that does not have any matching IPC information.

No Gerber data for location 1.5980, 4.3800 ID 45:() idx 43

There was an IPC-D-356 record for this location, but no Gerber data.

Gerber Net Re-assignment: VisualCAM net 78
Locations: 1.7980,0.8300 and 2.7980,4.2800
IPC nets 55:() 171:()

The IPC file has tried to associate the 2 nets, "55:()" and "171:()", to the VisualCAM net number 78

IPC Net Re-assignment: VisualCAM nets 123 250
Locations: 2.0980,1.0300 and 3.7980,4.3800
IPC net 78:()

The IPC file has tried to give the same net information "78:()" to the VisualCAM nets 123 and 250.

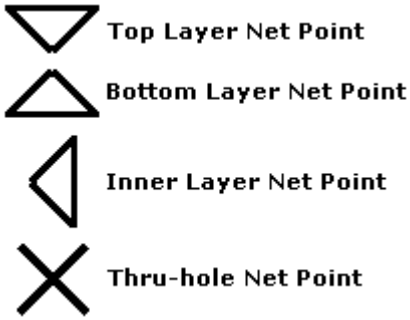
File|Import|Netlist|ODB++

As an aid to those who wish to compare a netlist created from Gerber data back to an ODB++ database netlist, the File|Import|Netlist|ODB++ command imports only the netlist from an ODB++ database. It is not necessary to use the File|Import|ODB++ command to import the entire database, export an IPC-D-356 netlist, discard the database, load your Gerber data and import the IPC-D-356 netlist.

If your goal is to perform a netlist comparison, see the *Comparing Netlists* tutorial for complete instructions. This topic will only cover the steps necessary for importing an ODB++ netlist.

Importing an ODB++ netlist

1. Select the File|Import|Netlist|ODB++ command. The Browse For Folder dialog box appears.
2. Pick the Job Folder that contains all your ODB++ database files, and click OK. The netlist is imported
3. After you import the netlist, a new External Net layer is created. Viewed by itself, this layer appears to contain no data. However, this layer contains the external netlist information, and links the external net points to the appropriate top/inner/bottom layer net test points. To view the points, turn on the associated top, bottom, and inner layers, and the external net layer. The net points are displayed in four ways for easy reference:



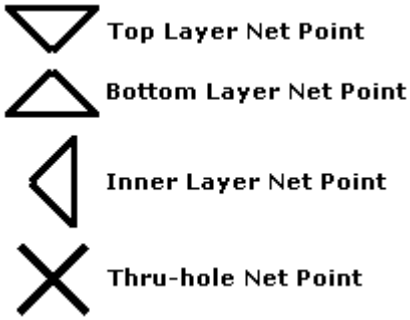
File|Import|Netlist|ODB++(X)

As an aid to those who wish to compare a netlist created from Gerber data back to an ODB++ XML database netlist, the File|Import|Netlist|ODB++(X) command imports only the netlist from an ODB++ XML database. It is not necessary to use the File|Import|ODB++(X) command to import the entire database, export an IPC-D-356 netlist, discard the database, load your Gerber data and import the IPC-D-356 netlist.

If your goal is to perform a netlist comparison, see the *Comparing Netlists* tutorial for complete instructions. This topic will only cover the steps necessary for importing an ODB++ XML format netlist.

Importing an ODB++(X) netlist

1. Select the File|Import|Netlist|ODB++(X) command. The Import ODB++(X) Netlist dialog box appears..
2. Select the ODB++ XML format database file, and click OK. The netlist is imported.
3. After you import the netlist, a new External Net layer is created. Viewed by itself, this layer appears to contain no data. However, this layer contains the external netlist information, and links the external net points to the appropriate top/inner/bottom layer net test points. To view the points, turn on the associated top, bottom, and inner layers, and the external net layer. The net points are displayed in four ways for easy reference:



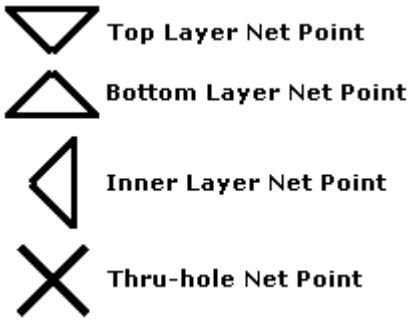
File|Import|Netlist|PADS ASCII

As an aid to those who wish to compare a netlist created from Gerber data back to a PADS ASCII database netlist, the File|Import|Netlist|PADS ASCII command imports only the netlist from a PADS ASCII database. It is not necessary to use the File|Import|PADS ASCII command to import the entire database, export an IPC-D-356 netlist, discard the database, load your Gerber data and import the IPC-D-356 netlist.

If your goal is to perform a netlist comparison, see the *Comparing Netlists* tutorial for complete instructions. This topic will only cover the steps necessary for importing a PADS netlist.

Importing a PADS netlist

1. Select the File|Import|Netlist|PADS ASCII command. The Import PADS ASCII Netlist dialog box appears.
2. Select your .asc file, and click Open. The netlist is imported
3. After you import the netlist, a new External Net layer is created. Viewed by itself, this layer appears to contain no data. However, this layer contains the external netlist information, and links the external net points to the appropriate top/inner/bottom layer net test points. To view the points, turn on the associated top, bottom, and inner layers, and the external net layer. The net points are displayed in four ways for easy reference:



File|Export

The Export commands are used for saving a database in various formats. If you wish to save the database in the VisualCAM .vcam format, use the File|Save command instead.

File|Export|Gerber

This command allows you to export one or more Gerber files from the current design.

To export Gerber files

1. Select the File|Export|Gerber command. The Export Gerber dialog box appears.
2. By default, files are exported to the Samples folder. If you would like your files exported to a different folder, click the Browse button next to the Destination Folder box, and find a different or new folder to export your files to.
3. A summary of the Gerber file format that will be used in your exported file is indicated next to the Data Format button. In this order, it shows the dialect, coordinate format, coordinate mode, units of measure, and zero suppression. If you want to change these or other format settings in your Gerber files, click the Data Format button. The Gerber Export Data Format dialog box appears, with the following options:

   Dialect
   This is the specific format of the Gerber file, either RS274D, RS274X, or FIRE9XXX.

   M.N
   This is the coordinate format, which specifies the decimal digits before and after the implied decimal point. Therefore, in a 2.4 format, 122550 would represent 12.2550. During export, the importance of the coordinate format is in the precision of your data. The more trailing digits (the "N"), the more precise your data will be. If you have a dense design, you should use a higher number of trailing digits.

Terminator
Indicate the desired block terminator (EOB). Usually this an asterisk. If desired, use **\r** to indicate a carriage return (ASCII 13), and **\n** to indicate a line feed (ASCII 10).

Apply To
These settings will be applied to all files that you select to export at this time.

Coordinate Mode
Absolute mode is when all X:Y coordinates are referenced to a common origin. Incremental mode is when each X:Y coordinate is a displacement from the previous coordinate.

Zero Suppression
No zero suppression allows coordinates to have zeroes in front or back of integers. Leading zero suppression shows zeroes only at the end of a number, trailing zero suppression shows zeroes only at the beginning of a number.

'G' Commands
Indicate whether G commands (i.e. G01) should be included when exporting Gerber files.

Character Set
Indicate the character text used in the file.

Special
You may select Metric mode to indicate that your files are in metric format. Otherwise the format will be Imperial (inches).

Modal mode reduces the size of your files by removing all redundant draft codes and coordinates.

The Optimize Polarity option only applies to exporting 274-X data. It minimizes the number of polarity changes in your Gerber files. For example, if Optimize Polarity is not selected, you could have several contiguous LPD or LPC commands in your Gerbers (these are commands that indicate dark or clear polarity). Other systems may read these commands and produce unnecessarily complex composite layer structures, instead of a single layer containing just one set of dark and clear data. If Optimize Polarity is selected, any contiguous LPD or LPC commands are combined into single LPD or LPC commands.

Select Comments to save G04 comments.

You can select to export User Data information, if you inserted any in your database.

If a netlist exists in your database, the Netlist option exports netlist information embedded within the Gerber files.

Extended Gerber
These options only apply if you are exporting 274-X data.

Selecting the Short 274-X Macro Names option exports all 274-X macro names using a 7 character limit. This setting is for compatibility with plotters that have limited support for the 274-X extended Gerber format.

Selecting the Include Unused Apertures and Macros option specifies that all aperture definitions (custom or otherwise) will be exported, whether they are used or not.

4. When you are finished defining your data format, click the OK button to return to the Export Gerber dialog box.
5. Select the layers you wish to export in the Filename column. For your reference, the associated layer's name and number are listed in the adjoining columns. Each layer type is represented by an icon. For an icon reference list, see the Color Bar topic. If you have a large number of layers, but only wish to export a few, click the Unselect All button, then individually select the desired layers. If you wish to change any of the file names, double-click on the filename you wish to change and then enter the new name.
6. When you are finished making your selections, click the OK button to export your files.
7. If you exported your files in the 274-D format, use the File|Export|Aperture List command to export the aperture list.

Related topics

Gerber for Beginners
Using Files

File|Export|Aperture List

This command allows you to export an Aperture List file from the currently loaded database. This file is only necessary if you are exporting your data as 274-D Gerber files.

To export an aperture list

1. Select the File|Export|Aperture List command. The Export Aperture List As dialog box appears.
2. Select a file name for your aperture list, and click the Save button. Only the VisualCAM .map format is supported, although you can assign any desired extension to the file.

File|Export|NC (Drill/Mill)

Toolbar Button: 

The Export NC (Drill/Mill) command creates an NC file for either drill data, mill data, or both, in the format specified.

To export NC files

1. It is recommended that you use the Analysis|DRC/MRC command to check for coincident and redundant drill hits, among any other desired checks, prior to exporting your data.
2. Use the Tools|NC|Optimize command to optimize the NC tooling.
3. Select the File|Export|NC (Drill/Mill) command. The Export NC dialog box appears.
4. Select a Destination Folder to place your NC files in. By default, the Samples folder is used. Click the browse button to search for a different folder.
5. The NC layers are listed at the bottom of the dialog box, with their layer names and numbers for your reference. In the Filename column, select (check) the layers you wish to export, and unselect those you do not want to export. You can change the file name by double-clicking on the name, and typing a new one.
6. The Data Format button opens the NC Export Format dialog box, which provides the following settings for your file formatting. Change these settings, as desired, and click the OK button to return to the Export NC dialog box.

   Dialect
   The format of the file to export. Available options are Excellon, Excellon 2, Sieb & Meyer 1000, Sieb & Meyer 3000/5000, and Takeuchi

   M.N
   This is the coordinate format. For example, a 2.3 format specifies two decimal digits before an implied decimal point and three following. (12250 represents 12.250 if the coordinate format is 2.3).

   Mode
   Absolute mode is when all X:Y coordinates are referenced to a common origin. Incremental mode is when each X:Y coordinate is a displacement from the previous coordinate.

   Zero Suppression
   No zero suppression allows coordinates to have zeroes in front or back of integers. Leading zero suppression shows zeroes only at the end of a number, trailing zero suppression shows zeroes only at the beginning of a number.

   Special
   You may enable Modal mode which will reduce the size of your files by removing all redundant draft codes and coordinates. Metric outputs the file in metric format. The default is imperial (inches).

   Character Set
   Indicate the character text used in the file.

   Set As Default
   Saves the current settings as the default settings for this command.
7. Select whether you wish to export Drill Data, Mill Data, or both, from your NC layers.
8. Select Use Step & Repeat to export panelized images, as well as any patterns that you defined in the NC Tool Table, as step and repeat data. If this option is not selected, then panelized images and patterns are exported individually (no step and repeat codes used). Use of step and repeat will reduce your file size.
9. Select Segment Arcs if your machine does not support arcs. This will convert true arcs in your mill paths to line segments.
10. In Excellon machine controllers, there is a "Tool Diameter Page" where information about a tool may be recorded. When the tool size in a tool definition is processed, the information (if any) from the Tool Diameter Page is loaded, and the location of the tool size in the definition string affects the tool settings. The Override Tool Diameter Page option controls where the system puts the tool size field in the tool definitions. If you select this option, the tool size is put immediately after the tool number, so any settings from the Tool Diameter Page will be overridden by those specified on the tool definition line. If this option is not selected, the tool size is the last entry in the tool definition line, so any settings in the Tool Diameter Page will override those in the tool definition. (Note that settings in the Tool Diameter Page are optional; they override or are overridden only if they exist.)
11. Tools are normally exported using the tool number defined in the NC Tool Table, and in the defined Export Order. If you select the Renumber Tools Starting With Tool option, the tools are exported in the Export Order, but the first tool number is changed to the number you specify here. Subsequent tools are renumbered, in sequence, accordingly.
12. Select the Export Unused Tools option if you wish to export all tools, regardless of if they have been used.

13. The default compensation values in the Compensation Index Table are set at the tool's radius, as that is assumed to be the machine's default. Therefore, if only default values are specified, the Compensation Index table is not exported with the rest of the NC data. However, if you want to always export the Compensation Index Table (because, for example, your machine does not default the compensation to the tool's radius), select the Always Export Compensation Index Table option.
14. When you are finished making your selections, click the OK button to export your files.

Related topics

Tools|Convert|Arcs To Segmented Arcs

File|Export|NC Tool List

This command allows you to export an NC tool list file in the VisualCAM .nct format. This is useful if you want to create a standard tool list for use in future jobs, as you can later use the Load button in the NC Tool Setup dialog box to open it.

To export an NC tool list

1. Select the File|Export|NC Tool List command. The NC Tool Setup dialog box appears.
2. In the Table list, select the table you wish to export the data for.
3. Click the Save button. The NC Tool Table Files dialog box appears.
4. Specify a name for the .nct file, and click Save.

File|Export|BARCO DPF

This command exports your layers to Barco .dpf files. A separate file is created for each layer.

To export in the Barco DPF format

1. Select the File|Export|BARCO DPF command. The Export Barco dialog box appears
2. In the Destination Folder specify the folder to save the .dpf files in. Use the Browse button to find the folder, if necessary.
3. The Filename column allows you to select the layers you wish to export, as well as specify the filename you wish to use for each layer. To unselect all layers, click the Unselect All button. To rename a file, double-click on the filename you wish to change and then enter the new name.

   Each layer type is represented by an icon. For an icon reference list, see the Color Bar topic. The Layer Name column displays the name of each layer. The Layer column displays each layer's number.
4. When you are finished, click the OK button. Your files are exported to the designated folder.

File|Export|IPC-D-350

Use this command to export your design to an IPC-D-350 file. A single .ipc file will maintain all your layer data in an "intelligent" format.

To export an IPC-D-350 file

1. Select the File|Export|IPC-D-350 command. The Export IPC-D-350 dialog box appears.
2. Specify a file name, and click the Save button. Your database is now saved.

File|Export|IPC-D-356

This command creates a single IPC-D-356 or 356A format file with the netlist data for the currently loaded design, according to your specifications. If you have reverse engineered your data for assembly, the reference designator and pin number information is automatically output in the file.

To export an IPC-D-356 netlist

1. If your database does not already have a netlist, use the Tools|Netlist|Generate command to create one.
2. Select the File|Export|IPC-D-356 command. The Export IPC-D-356A dialog box appears.
3. In the File Name box, enter a name for your netlist file. By default, the file is saved in VisualCAM's Samples folder. Use the Browse button to choose a different folder or file name.
4. Enter the number of the NC layer that contains your through-hole drill information (this is the same NC layer that was used when the netlist was generated). If no NC layer is available, the system will look at the aperture table "type" field to determine whether a pad is SMT or not.
5. Select Include Unconnected Pads Using N/C Net Name to have all unconnected (isolated) pads output using the IPC-defined net name "N/C". Otherwise, all unconnected pads will be ignored.

6. Select Include Solder Mask Information to include data indicating solder mask usage. If no solder mask is present, then this option is not available. (If you have a solder mask but this option is not available, your solder mask layer is not properly tagged in the layer table.)
7. If you have reverse engineered part data for assembly, select the Export Component Information option to include reference designators and pin numbers in your IPC-D-356A file.
8. *You do not need to select any of the other options in the dialog box.* If they are selected, unselect them. They apply only to IPC-D-356A netlists.
9. Click the OK button to export your file.

To export an IPC-D-356A netlist

1. If your database does not already have a netlist, use the Tools|Netlist|Generate command to create one.
2. Select the File|Export|IPC-D-356 command. The Export IPC-D-356A dialog box appears.
3. In the File Name box, enter a name for your netlist file. By default, the file is saved in VisualCAM's Samples folder. Use the Browse button to choose a different folder or file name.
4. Enter the number of the NC layer that contains your through-hole drill information (this is the same NC layer that was used when the netlist was generated). If no NC layer is available, the system will look at the aperture table "type" field to determine whether a pad is SMT or not.
5. Select Include Unconnected Pads Using N/C Net Name to have all unconnected (isolated) pads output using the IPC-defined net name "N/C". Otherwise, all unconnected pads will be ignored.
6. Select Include Solder Mask Information to include data indicating solder mask usage. If no solder mask is present, then this option is not available. (If you have a solder mask but this option is not available, your solder mask layer is not properly tagged in the layer table.)
7. Normally, only top and bottom net locations are exported. Select the Export Inner Layer data option to include inner layer net locations, which is most often used if your design has blind and buried vias.
8. The Export Conductor Data option exports the trace data for the top and bottom layers. If you selected the Export Inner Layer Data option, then the inner layer trace data are also included.
9. IPC-D-356A does not support raster fill polygons. If you want polygon information maintained with your trace data, select the Vector Fill Polygons And Export With Conductor Data option to make the polygons vector fill. Also specify the size of the lines to use for the solid vector fill.
10. Export Net To Net Adjacency Data to report any nets within your specified distance to each other. (In the netlist, each net is reported along with any other net that is adjacent to it, according to the distance you specify.) Select this option, if desired.

⚠ *Tip: The Export Probe Points option is only available when exporting flying probe test data.*

11. If you have reverse engineered part data for assembly, select the Export Component Information option to include reference designators and pin numbers in your IPC-D-356A file.
12. Click the OK button to export your file.

File|Export|IPC-2581 (Offspring)

This command is used to export IPC-2581 (aka the Offspring format) files.

To export an IPC-2581 database

1. Select the File|Export|IPC-2581 (Offspring) command. The Export Offspring Design dialog box appears.
2. Specify a file name for your exported .cvg file, and click the Save button. Your file is exported.

File|Export|GenCAD

This command is used to export the current database to a GenCAD Design file.

To export a GenCAD database

1. Select the File|Export|GenCAD command. The Export GenCAD Design dialog box appears.
2. Specify a file name for your exported .cad file, and click the Save button. Your file is exported.

File|Export|DXF

This command allows you to export your VisualCAM design to one or more DXF files (as desired). Each VisualCAM layer creates a corresponding DXF layer. In addition, a DXF layer 0 is created containing items which appear within the Blocks section. The Blocks section contains information necessary for displaying each of the apertures used in the design. Block names are created with a convention that allows for easy import of aperture information back into VisualCAM (using DXF Import's Block Mapping function). Each pad in the design is output into the DXF file as a block

insert. Outputting the pads as references in this manner instead of duplicating the draws for each instance can significantly reduce the size of the generated file.

To export a DXF file

1. Because DXF does not support the concept of polarity, items with clear or negative polarity will not appear correctly when the file is imported into other applications. If you have items with negative (clear) polarity in your database, run the Tools|Convert|Composite To Layer command to convert everything to positive (dark) data.
2. Select the File|Export|DXF command. The Export DXF dialog box appears.
3. In the Output File box, specify the filename you wish to use for the DXF file. By default, the file is saved in VisualCAM's Samples folder. Use the browse button if you wish to save to a different folder, or find a file to overwrite.
4. The export function produces a report file, which contains a summary of the DXF file: the number of layers, flash inserts, lines, polygons, and blocks. Enter a name for the .txt file in the Report File box.
5. Select Metric to have the coordinates in the DXF file expressed in millimeters. Otherwise, coordinates are in inches.
6. Select Batch Mode to output each visible layer in a design to a separate file. If the Output File box is empty, the output filenames will be derived from each layers filename and the current output filename extension. If the output field contains a name, a number is appended, representing each layer's respective number.
7. Select Output Endcaps to have endcaps output for each draw in the DXF file.
8. Select Round Pads As Circles to export round pads as unfilled circles. Otherwise, pads are exported as "solid" entities using arcs.
9. If you have polygons, select the way in which you want to output them. Polyline outputs the polygons as with just an outline. Hatch outputs the polygons with a solid "raster" fill. Vector Fill outputs the polygons with a vector fill, where you specify the width of the vector fill line in the Size box.

File|Export|GerbTool

This function allows you to export your design data to a GerbTool version 15.X database (.gtd) file.

To export a GerbTool database

1. Select the File|Export|GerbTool command.
2. Select a name for your .gtd file, and click the Save button.

Related topics

File|Save

File|Export|ODB++

This command exports uncompressed ODB++ database files.

To export an ODB++ database

1. Select the File|Export|ODB++ command. A Browse For Folder dialog box appears.
2. Pick the desired Job Folder to export your data to, or click Make New Folder to create that new one.
3. Click the OK button, and the files are exported.

File|Export|ODB++(X)

This command exports ODB++ database files in the XML format.

To export an ODB++(X) database

1. Select the File|Export|ODB++(X) command. The Export ODB++(X) Design dialog box appears.
2. Browse to the folder you wish to place the file in, and specify a file name for it.
3. Click the Save button, and the file is exported.

File|Export|HPGL

This command creates an HPGL plot file, or prints to an HPGL printer. There are two files within the VisualCAM program directory that affect each HPGL plot. The files hpgl.ini and hpgl.dei are prefixed and appended, respectively, to the actual plot output. If you have any special requirements, you may edit these files as needed.

To export HPGL files

1. Only visible layers will be exported. Turn on the visibility of any layers you wish to include, and turn off any layers you wish to exclude.
2. Select the File|Export|HPGL command. The Export HPGL dialog box appears.

3. In the Output File box, select a file name for your PostScript file. By default, the file is saved in VisualCAM's Samples folder. Use the browse button if you wish to save to a different folder, or find a file to overwrite.

   -or-

   To send the output to a plotter, specify output port (e.g. "LPT1:" or "COM2:") instead of a file name.
4. If you wish to increase or decrease the size of the plot, select a scale for the exported data. A value of 1.0 plots at 100%, 0.5 at 50% etc.
5. The Media Size is the size of the printable area of the printer's output media, in whatever units of measure you have currently defined (see the Options|Units/Precision command if you want to view or change the units of measure).
6. If you wish to change the positioning of the output on the page, change the Offset values. The offset values are applied independent of the scale specified.
7. The Plot Size options allow you to select the style of plotter being used. There are two main types. Generally, A/B style plotters have their plotter origin located in the lower left corner of the page, while the C/D/E style has their origin in the center of the page.

Some plotters do not follow the above convention for locating their origins. If you are experiencing difficulties in plotting, check your plotter manual to determine the location of their origin and set the VisualCAM parameters accordingly.

8. The Mode setting selects the style of plotter output.

   Sketch mode is the fastest but does not show width on draws and some flashes, such as Donuts.

   Outline mode shows true width on all objects but they are outlined only.

   Fill mode shows true width and all objects are completely filled in. Fill mode is the slowest to plot and is extremely hard on plotter pens.
9. If you wish to rotate the plot 90 degrees counter-clockwise, select the Rotate option.
10. Enabling Interactive mode allows you to interactively position each layer on the output page.
11. Select the Pads Only option if you want only flashed pads to be plotted.
12. To output each visible layer to a separate output file, select Batch Mode. If the Output File box is empty, the output filenames will be derived from each layers filename and the current output filename extension. If the Output File box contains a name, a number representing the layer number of each layer plotted will be appended to the name.
13. To have a border drawn around the plot, select the Add Border option. The Border Pen option controls which pen is used for adding the border.
14. Click the Setup Pens button. The HPGL Pen Setup dialog box appears. Here, you can designate different plotter pens for draws and flashes on a per-layer basis. To change a pen assignment, click on the desired layer in the Layers list. In the boxes below, change the pen information as desired. After making your changes, select another layer, or click the OK button to return to the Export HPGL dialog box.
15. Click OK to plot or save your file.
16. If you selected Interactive mode, the workspace alters to show a representation of each layers outline with its corresponding number. To position an image on the page, simply click on an image to select it and then drag the image to the desired location and release the mouse button. During interactive plot positioning, a button bar and with several plot-specific hotkeys are provided as an aid. The hotkeys available during an interactive plot session are **C** for absolute coordinate entry, **I** for page layout initialization, **L** to cycle the currently selected layer forward, **Ctrl+L** to cycle the currently selected layer backward, **S** to snap (align) the currently selected layer on top of another layer, and **R** to redraw the page layout. When you are finished, click the OK button in the button bar.

Related topics

File|Print

File|Export|PostScript

This function is used to create PostScript files, or to plot data on any PostScript printer. This includes typesetters capable of producing production quality artwork.

To export a PostScript file

1. Only visible layers will be exported. Turn on the visibility of any layers you wish to include, and turn off any layers you wish to exclude.
2. Select the File|Export|PostScript command. The Export PostScript dialog box appears.
3. In the Output File box, select a file name for your PostScript file. By default, the file is saved in VisualCAM's Samples folder. Use the browse button if you wish to save to a different folder, or find a file to overwrite.

-or-

To send the output to a plotter, specify output port (e.g. "LPT1:" or "COM2:") instead of a file name.

4. If you wish to increase or decrease the size of the plot, select a Scale for the exported data. A value of 1.0 plots at 100%, 0.5 at 50% etc. If you want to automatically scale the plot to fit to your page (the Media Size), select the Fit To Page option.
5. The Media Size is the size of the printable area of the printer's output media, in whatever units of measure you have currently defined (see the Options|Units/Precision command if you want to view or change the units of measure).
6. If you wish to change the positioning of the output on the page, change the Offset values. The offset values are applied independent of the scale specified.
7. Select one of the two Modes of output. Sketch mode shows true width on all objects but they are outlined only. This is the fastest method and allows you to check for overlapping features, but does not show width on draws and some flashes, like Donuts. Fill mode shows true width and all objects are completely filled in as they would appear on a plot. Fill mode may produce a larger output file.
8. Select the Pads Only option if you want only flashed pads to be plotted.
9. Selecting the Gray Scale option outputs all colors (except black and white) in shades of gray. When Gray Scale mode is not selected, all data are displayed as black on a white background.
10. If you wish to rotate the plot 90 degrees counter-clockwise, select the Rotate option.
11. To have a border drawn around the plot, select the Add Border option.
12. To output each visible layer to a separate output file, select Batch Mode. If the Output File box is empty, the output filenames will be derived from each layers filename and the current output filename extension. If the Output File box contains a name, a number representing the layer number of each layer plotted will be appended to the name.



File|Print

File|Export|Bitmap

This command allows you to export the current view to a Windows .bmp file. This is the standard bitmap file format that virtually all graphics and page layout applications use.

To export a bitmap file

1. Only visible layers will be exported. Turn on the visibility of any layers you wish to include, and turn off any layers you wish to exclude.
2. If you plan to export the bitmap using color, use the Color Bar to set your layers, Options|Configure/Display feature to set the workspace background, to your desired colors.
3. Select the File|Export|Bitmap command. The Export Bitmap dialog box appears.
4. In the Output box, specify the filename you wish to use for the bitmap file. By default, the file is saved in VisualCAM's Samples folder. Use the browse button if you wish to save to a different folder, or find a file to overwrite.
5. Specify the Resolution you wish to use for the bitmap, in dots per inch (DPI).
6. Select the Black & White if you wish to have the bitmap file created with only black and white colors. Otherwise it will be output in color.
7. If you select Black & White, specify the color of the background you wish to have used for the output bitmap file. If you choose a black background, your layer data will show as white. If you choose a white background, your layer data will show as black.
8. When you have finished making your selections, click the OK button. Your file is exported.

File|Page Setup

This command allows you to configure each page printed by the Print command. This does not affect the output for any of the Export functions, such as HPGL or PostScript.

The Page Setup dialog box presents the following options:

Scale
This is used to change the scale of the output. A value of 1.0 plots at 100%, 0.5 at 50%, etc.

Overlap
If the resulting output will span more than one page, pages will overlap by this amount. This makes it easier to align the pages to produce one larger plot.

Border Text
Specify the border text you wish to use by entering any of the following keywords:

$TIME displays the time.
$DATE displays the date.
$DESIGN displays the .vcam file name.
$LAYERNAME displays the layer name
$PROG displays the "VisualCAM" program name.
$SCALE displays the scale factor.

To display this border information, make sure you select the Add Border option below.

Add Border
Select Add Border to place a border around the plot, with any specified Border Text appearing at the bottom.

Batch Mode
Select Batch Mode to have each visible layer in a design is plotted separately, on its own page(s).

Color/Grayscale
Select Color/Grayscale to produce a plot using the color or grayscale capabilities of the printer. If it is not selected, the output will be forced to black on a white background.

Fit To Page
Select Fit To Page to scale the plot to fit the current page size

Center
Select Center to center the plot on the page (when the plot fits entirely on one page).

Overlay
Select Overlay to have overlapping areas of draws and flashes plotted in different colors. This option is most useful when plotting multiple layers.

Print Background
Select Print Background to have the background appear in the current workspace background color. Otherwise the background will be white.

Sketch
Select Sketch to plot everything in sketch mode. If not selected, items are plotted in their normal, filled appearance.

Window
Select Window to only print what is contained within a windowed area (that you define at the time of printing).

Screen
Select Screen print what is displayed in the current view window instead of the extents of the entire view design.

Set As Default
Saves the current settings as the default settings for this command.

Related topics

File|Print
File|Print Setup
File|Print Preview

File|Print

Toolbar Button: 

The Print command prints the currently visible layers. VisualCAM uses the standard Windows printing utility, and can plot to any printer supported by Windows.

To define the format of each page, use the File|Page Setup command.

If you wish to create an HPGL or PostScript file, or print to an HPGL or PostScript printer, use the appropriate Export function.

Related topics

File|Print Setup
File|Print Preview

File|Print Preview

This command allows you to view how each page of your design would print on the current Windows printer, before you actually print it.

Related topics

File|Page Setup
File|Print Setup
File|Print

File|Print Setup

This command allows you to select and configure the printer prior to using the Print command. VisualCAM uses the standard Windows printing utility, and can plot to any printer supported by Windows.

Related topics

File|Page Setup
File|Print
File|Print Preview

File|Send

This command allows you to quickly send the currently loaded database to an e-mail recipient.

To send a database by e-mail

1. Select the File|Send command. Your default e-mail program automatically creates a blank e-mail, with the currently loaded design file attached to it (in the .vcam format).
2. Specify the recipient's e-mail address, and the e-mail's subject. You can also attach additional files or add text to the e-mail's body, as desired.
3. Send the e-mail.

File|Save Log

The Log screen, which records your actions, can be saved as a .txt file by selecting the Save Log command. You can also right-click anywhere in the Log screen, and select Save Log from the shortcut menu.

MRU File List

This is a list of the most recently used design files. To reload one of these designs, simply select it from the menu.

File|Exit

Equivalent Hotkey: Ctrl+ALT+Q (quit immediately without file save confirmation)

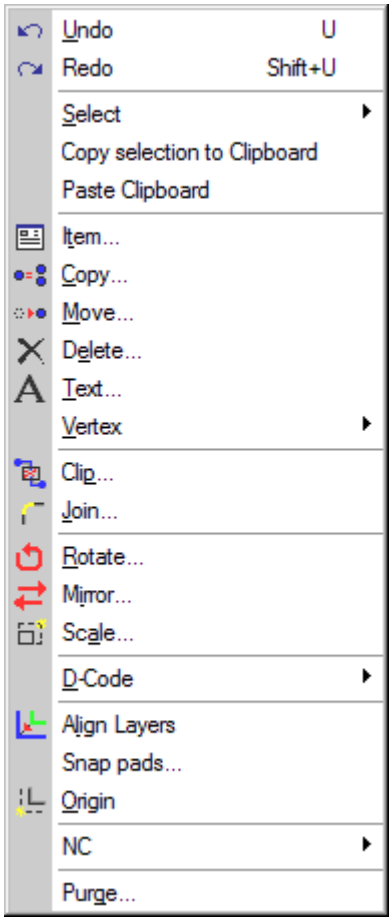
This command closes the VisualCAM application. If you only wish to close the current database, and continue to work on a different database with VisualCAM, use the File|Close command.

To exit VisualCAM

1. Select the File|Exit command.
2. If the current design file has not been saved, you are asked whether you wish to save it now. If you do not save it, any changes you made since you last saved the file will be discarded.

Edit menu

The Edit menu presents you with commands that modify one or more database items.



Edit|Undo

Toolbar Button: 

Equivalent Hotkey: U (undo last edit) or **Ctrl+U** (undo all edits)

The Undo command reverses changes you have made to the currently loaded database. Note that some functions cannot be undone; the Undo command is only available when a function allows it.

If you plan to use the Undo command, it must be enabled with the Options|Configure command prior to making any edits. Undo increases the amount of memory VisualCAM requires. If you do not require the undo capability, you may disable it. Disabling undo releases any memory currently associated with undo information, and prevents further memory use.

To undo a change

1. Select the Edit|Undo command. The Undo dialog box appears, with your last seven edits listed (the most recent edit being listed first).
2. In order to undo one change, you must undo all subsequent changes (e.g. if you add a draw, then a flash, and you wish to undo the draw, you must undo the flash as well). Select the change(s) you wish to undo in the dialog box.
3. Click the Undo button. The dialog box closes, and the workspace is immediately updated.

Related topics

Edit|Redo

Edit|Redo

Toolbar Button: 

Equivalent Hotkey: Shift+U (reverse last undo)

Where the Undo command reverses changes you have made to your database, the Redo command reverses the Undo command. Your database is returned to the state it was in prior to your selection of the Undo command.

If you plan to use the Undo & Redo commands, the Undo command must be enabled with the Options|Configure command prior to making any edits. Undo increases the amount of memory VisualCAM requires. If you do not require the undo capability, you may disable it. Disabling undo releases any memory currently associated with undo information, and prevents further memory use.

To redo a change

1. Select the Edit|Redo command. The Redo dialog box appears with the last seven "undone" edits listed.
2. You redo changes in the order that they were originally performed, and in order to redo one change, you must redo all changes made prior to it. Select the change(s) you wish to redo in the dialog box.
3. Click the Redo button. The dialog box closes, and the workspace is immediately updated.

Edit|Select

Most Edit commands (Copy, Move, etc.) allow you to work with single items, windows of items, or groups of items. The Select commands allow you to manage groups of items to be modified by the Edit commands.

When a command allows group selection mode, it will use the current Select Group that you created using the different Select commands. Select Groups are also persistent from one Edit command to another. For example, if you rotate the current Select Group, the rotated data will remain selected, ready for another command.

Edit|Select|New Group

Toolbar Button: 

The New Group command allows you to create a new Select Group items. Once you have created a select group, the group is added as a Mode in the Selection Filter, for use future Edit functions.

Tip: If you want to add the majority of the items in a database to the Select Group, you can use this function to first create a Select Group of the items you ultimately wish to exclude. Then select the Edit|Select|Invert command to reverse the selection, and create your desired Select Group.

To create a Select Group

1. Select the Edit|Select|New Group command. If you already had a Select Group defined, you are asked if you wish to clear the current group in favor of a new group. Doing so does not delete or otherwise alter any data.
2. Using the Selection Filter as necessary, add any desired items to your Select Group.
3. When you are finished adding items, press the Esc key to exit the function. Your selections are automatically saved, and will remain highlighted in the workspace. (To turn off the highlighting, select the View|Highlights command).

If you later wish to add items to the Select Group, use the Edit|Select|Add To command. If you wish to remove items from the group, use the Edit|Select|Remove From command.

Edit|Select|Add To

Toolbar Button: 

Use the Add To command to add additional items to the current Select Group. If you do not already have a Select Group created, this will create a new one.

To add to the current Select Group

1. Select the Edit|Select|Add To command.
2. Using the Selection Filter as necessary, add any desired items to your Select Group.
3. When you are finished adding items, press the Esc key to exit the function. Your selections are automatically saved, and remain highlighted in the workspace. (To turn off the highlighting, select the View|Highlights command).

Related topics

Edit|Select|New Group

Edit|Select|Remove From

Toolbar Button: 

Use the Remove From command to selectively remove items from the current Select Group. It does not delete or otherwise alter any data -- the item is only removed from the group, and it is no longer highlighted in the workspace.

If you would like to remove all items from the Select Group, use the Edit|Select|Clear command.

To remove items from the Select Group

1. Select the Edit|Select|Remove From command.

2. Using the Selection Filter as necessary, select the items you want to remove from the Select Group.
3. When you are finished, press the Esc key to exit the function.

Edit|Select|Invert

Toolbar Button: 

Use this command to invert the current Select Group. That is, all currently items in the Select Group are removed from the group, and all other items in the database are selected.

If you are creating a select group from the majority of the items in your database, you can quickly create the group by first selecting the items you do not want with the Edit|Select|New Group command, and then inverting the group selection with this command.

⚠ *Warning: This command will select items from both visible and invisible layers.*

Related topics

Edit|Select|Remove From

Edit|Select|Clear

Toolbar Button: 

The Clear command empties the current Select Group and removes highlighting of the Select Group items in the workspace. It does not delete or otherwise alter any data -- the items are only removed from the group, and are no longer highlighted in the workspace.

If you only wish to remove certain items from the Select Group, use the Edit|Select|Remove From command.

When you select the command, you are asked to confirm if you wish to clear the Select Group. After you click the Yes button, the group is immediately cleared.

Edit|Copy Selection To Clipboard

Equivalent Hotkey: Ctrl+I (copy entire workspace to Windows Clipboard)

This command copies selected items to the Windows Clipboard, allowing you to copy items from one database to another. It also allows you to copy entire sections of a design from one area and paste them to another area of the same database. The layer information is maintained, even if you copy from multiple layers. If you want to copy items from one layer to another, use the Edit|Copy command. To copy all the data in a layer to another layer, use the Layers list in the Navigator.

⚠ *Tip: The Copy Selection To Clipboard command itself does not allow you to copy select items into other applications. You can copy the entire workspace to the Windows Clipboard using the Ctrl+I hotkey, however, which allows you to paste into other applications.*

To copy items using the Clipboard

1. Use the Edit|Select|New Group command to create a Select Group.
2. Select Edit|Copy Selection To Clipboard, and your selections are immediately copied. If you are pasting the information to another database, you may now close your current database and open another one.
3. Paste the items using the Edit|Paste Clipboard command.

Edit|Paste Clipboard

If you have copied items from another VisualCAM database to Windows Clipboard, this command pastes the items back into VisualCAM. See the instructions for the Edit|Copy Selection To Clipboard command for more details.

⚠ *Warning: This command does not allow you to copy items from other applications.*

Edit|Item

Toolbar Button: 

This command allows you to edit pertinent information associated with each database item. In addition to allowing you to edit each database item, there are extensive controls for navigating from one item to another. This includes the ability locate an item based on its aperture, net, UserData value, etc.

To edit a database item

1. Select the Edit|Item command. The Edit Item dialog box appears.
2. Using the Selection Filter, select the item you wish to modify. The Edit Item dialog box displays information about the item, and allows you to edit it, using the following controls:

Seq No
Displays the sequence number of the item whose information is being displayed. The sequence number is an item's approximate location inside the database. You cannot modify an item's sequence number. If you know an items sequence number, however, you can enter it into the Seq No box and click the GoTo button. The appropriate item will be found in the database, and its information displayed below. You can also click the << or >> buttons to display the preceding or succeeding items, respectively.

Delete Item
To delete the item currently displayed in the Edit Item dialog box, select the Delete Item option and click the Apply button.

Layer
The Layer box displays the layer the currently selected item is displayed on, with the layer's name displayed next to it. To move the item to a different layer, change the layer number (you can also select a layer from a list by clicking the arrow button). Then click the Apply button.

D-Code/Tool
The D-Code box displays the number of the D-code used to create the selected item, and the D-code's properties are displayed next to it. To change an item's D-code, change the D-code number (you can also select a D-code from a list by clicking the arrow button). Then click the Apply button. To highlight each individual item that uses the selected D-code, click the Find button. (You can continuously click the Find button to cycle through all items.)

If the item you have selected is on an NC layer, the tool that is used for that item is displayed instead. You can change or find the tool assignment just like a D-code.

XY
This box displays the coordinates of the currently selected object. To change an item's coordinates, change the values and click the Apply button. To find all items at a particular set of coordinates, specify the coordinates and click the Find button. (You can continuously click the Find button to cycle through all items.)

Polarity
This displays the polarity of the selected item. To change an item's polarity, select the desired polarity and click the Apply button.

⚠ *Warning: If you plan to export your data in the 274-D or DXF format, you should not change items to Clear polarity. 274-D and DXF do not support polarity.*

Select Group
This box will be checked if the selected item is currently a member of the current Select Group. Changing the status of this check box and pressing the Apply button modifies the select group status of the current item.

Net
This is the net number of the selected item. To find all items that comprise a particular net, type the Net number in the box and click the Find button. (You can continuously click the Find button to cycle through all items.)

⚠ *Warning: If you change the net number for an item and press the Apply button, you will change the net that it belongs to. This is not recommended, as this can cause errors in your netlist.*

UserData
This is the user data that has been associated with the selected item. If you place information into this area and press the Apply button, the new data will be associated with the displayed item. To find all items that have specific user data already assigned to it, type in the user data and click the Find button.

The UserData field allows you to attach text to each individual database item. Any user data you associate with your design will automatically be saved when you save your database. This allows you to pass on this data to other groups in your organization transparently.

User data can be used to associate reference designators, pin numbers and net names with each pad, thereby adding intelligence to your Gerber databases. Besides being able to see UserData using the Query|Item command, Macros also have complete read/write access to each UserData field.

Other than a 256 character limit, there are no other restrictions on what text can be associated with a database item.

Undo
Clicking the Undo button reverses the last change made with this command.

Redraw
Clicking the Redraw button redraws the data in the workspace, so you can see the effect of your changes.

Reset
Clicking the Reset button causes the data in this form to be reset to its initial values, removing any changes you may have specified.

Tip: Once the Apply button has been pressed, the Reset button will only revert back to the last applied state (it will not undo applied changes). Use the Undo button to reverse any applied changes.

3. When you are finished, click the Close button.

Edit|Copy

Toolbar Button: 

You may use this command to copy single items, windowed selections, or a Select Group of items.

To copy items from one database to another, use the Edit|Copy Selection To Clipboard command. To copy all the data in a layer to another layer, use the Layers list in the Navigator.

If you copy drawn or flashed data from a graphics layer to an NC layer, the system will convert all draws to mill paths, and all flashes to drill hits. If tools the same size as the draws/flashes do not exist in the tool table that is assigned to the NC layer, new tools will be created. You can also use the Tools|Convert|Gerber To NC command to copy and convert an entire graphics layer to an NC layer.

To copy a single item

1. Select the Edit|Copy command.
2. In the Selection Filter, select Item mode, as well as the type of item that you wish to select.
3. Click on the item you wish to copy in the workspace. If more than one item exists at that location, the Choose Selection dialog box appears, where you may select the desired item.
4. An outline of the item is attached to your cursor. Move your cursor to the desired location for the copy, and click to place it.

   -or-

   You can select exact coordinates for your placement in the XY Bar.
5. If you wish to place additional copies of the same item in other locations, click on those locations. Otherwise, press the Esc key.
6. You can now select another item to copy, or press the Esc key to end the function.

To copy a windowed selection of items

1. Select the Edit|Copy command.
2. In the Selection Filter, select Window mode if you wish to copy items that fall entirely within your designated selection window. Select Window +Xing mode if you want to copy any item that crosses into your designated selection Window. Also choose the type of item that you wish to select.
3. In the workspace, click on the lower-left corner of the desired selection area. The selection window is now attached to your cursor.
4. Move your cursor, and click on the upper-right corner of the desired selection area. The selected items are highlighted, and you are prompted to select a "copy from point."
5. Click on an anchor point, where the items will be attached in relation to your cursor. An outline of the data being copied is attached to your cursor.
6. Move your cursor to the desired location for the copy, and click to place it.

   -or-

   You can select exact coordinates for your placement in the XY Bar.

Tip: If you select data from more than one layer, i.e. more than one layer is visible, then the data will be copied into their respective source layers.

7. If you wish to place additional copies of the same items in other locations, select those locations. Otherwise, press the Esc key.
8. You can now select other items to copy, or press the Esc key to end the function.

To copy a Select Group

1. Create a Select Group using the Edit|Select|Add To command.
2. Select the Edit|Copy command.
3. In the Selection Filter, select Group mode. All other types of filters are not applicable.
4. Click on an anchor point, where the items will be attached in relation to your cursor. An outline of the data being copied is attached to your cursor.
5. Move your cursor to the desired location for the copy, and click to place it.

-or-

You can select exact coordinates for your placement in the XY Bar.

Tip: If your Select Group contains items from more than one layer, then the items will be copied into their respective source layers.

6. If you wish to place additional copies of the same items in other locations, select those locations. Otherwise, press the Esc key.
7. Press the Esc key to end the function.

To copy items to another layer

1. After having selected your items using one of the methods above, and when they are attached to your cursor, click on the Destination Layers button in the Status Bar. The Copy Destination Layers dialog box appears.
2. Select one or more destination layers where you want to copy to. Copies of all the selected data (regardless of their layer of origin) will be merged into each selected destination layer.
3. Move your cursor to the desired location for the copy, and click to place it.

   -or-

   You can select exact coordinates for your placement in the XY Bar.
4. If you wish to place additional copies of the same items in other locations, click on those locations. Otherwise, press the Esc key.
5. You can now select other items to copy, or press the Esc key to end the function.

Edit|Move

Toolbar Button: 

You may use the Move command to move a single item, a window, or groups of items.

If you wish to move items in an NC object, such as a mill path segment, use the Edit|NC|Explode command first to explode the object into its individual base elements. In the case of a mill path, it is converted into several mill paths, one for each segment. When you are finished, you can then use the Edit|NC|Combine Paths command to rejoin the segments.

To move a single item

1. Select the Edit|Move command.
2. In the Selection Filter, select Item mode, as well as the type of item that you wish to select.
3. Click on the item you wish to move in the workspace. If more than one item exists at that location, the Choose Selection dialog box appears, where you may select the desired item.
4. An outline of the item is attached to your cursor. Move your cursor to the new desired location, and click to place it.

   -or-

   You can select exact coordinates for your placement in the XY Bar.
5. Select another item to move, or press the Esc key to end the function.

To move a windowed selection of items

1. Select the Edit|Move command.
2. In the Selection Filter, select Window mode if you wish to move items that fall entirely within your designated selection window. Select Window +Xing mode if you want to move any item that crosses into your designated selection Window. Also choose the type of item that you wish to select.
3. In the workspace, click on the lower-left corner of the desired selection area. The selection window is now attached to your cursor.
4. Move your cursor, and click on the upper-right corner of the desired selection area. The selected items are highlighted, and you are prompted to select a "move from point."
5. Click on an anchor point, where the items will be attached in relation to your cursor. An outline of the data being moved is attached to your cursor.
6. Move your cursor to the desired location for the move, and click to place it.

   -or-

   You can select exact coordinates for your placement in the XY Bar.

Tip: If you select data from more than one layer, i.e. more than one layer is visible, then the data will be moved into their respective source layers.

7. Select other items to move, or press the Esc key to end the function.

To move a Select Group

1. Use the Edit|Select|New Group command to create a Select Group.
2. Select the Edit|Move command.
3. In the Selection Filter, select Group mode. All other types of filters are not applicable.
4. Click on an anchor point, where the items will be attached in relation to your cursor. An outline of the data being moved is attached to your cursor.
5. Move your cursor to the desired location for the move, and click to place it.

   -or-

   You can select exact coordinates for your placement in the XY Bar.

Tip: If your Select Group contains items from more than one layer, then the items will be moved into their respective source layers.

6. Press the Esc key to end the function.

To move items to another layer

1. After having selected your items using one of the methods above, and when they are attached to your cursor, click on the Destination Layer button in the Status Bar. The Move dialog box appears.
2. Select the destination layer where you want to move the items to. All the selected data (regardless of their layer of origin) will be moved to the selected destination layer.
3. Move your cursor to the desired location for the move, and click to place it.

   -or-

   You can select exact coordinates for your placement in the XY Bar.
4. Select other items to move, or press the Esc key to end the function.

Related topics

Tools|NC|Drill Separate

Edit|Delete

Toolbar Button: 

Select this command when you wish to delete items from one or more visible layers. If you wish to delete items from an NC object, such as a mill path segment, use the Edit|NC|Explode command first to explode the object into its individual base elements. In the case of a mill path, it is converted into several mill paths, one for each segment. You can then join the remaining segments back together with the Edit|NC|Combine Paths command.

You can also use the Edit|Clip command to delete windowed sections from a draw or mill path.

To delete items

1. Select the Edit|Delete command.
2. Select an item or items to delete. Use the Selection Filter to aid your selections, as necessary
3. Each time you select an item for deletion, you are prompted for confirmation. If you do not want to be prompted, click the Disable Prompts button in the Status Bar.
4. When you are finished, press the Esc button.

Edit|Text

Toolbar Button: 

This command edits existing text in a database; either the drilled text that resides on NC layers, or the standard text that resides on other layer types.

Tip: This function is also available through the right-click shortcut menu.

If the text you wish to edit is redline text, use the Documentation|Redline|Delete command to remove the text, then replace it with one of the appropriate Documentation|Redline commands. If the text you wish to edit is a fabrication note balloon, use the Documentation|Drawing|Fabrication|Modify Note Balloon command.

To edit standard text

1. Select the Edit|Text command.
2. In the Selection Filter, make sure that Text is selected in the Types list, and the layer the text is on in the Layers list.
3. Click on the text you wish to edit. The Modify Text dialog box appears. See the Add|Text command for information about text parameters.
4. Modify the text as necessary, and click Apply. The text in the workspace is updated immediately.

To edit drilled text

1. Select the Edit|Text command.
2. In the Selection Filter, make sure that Drilled Text is selected in the Types list, and the NC layer the drilled text is on in the Layers list.
3. Click on the drilled text you wish to edit. The Modify Drilled Text dialog box appears. See the Add|Drilled Text command for more information on drilled text parameters.
4. Modify the text as necessary, and click OK. The text in the workspace is updated immediately.

Related topics

Edit|Delete
Edit|Copy

Edit|Vertex

The Vertex commands are used to manipulate line vertices in your database.

Edit|Vertex|Add

Toolbar Button: 

This command allows you to add a vertex anywhere on an existing line segment or mill path.

Tip: The current orthogonal line snap setting affects this function (if the Ortho Line Snap is turned on).

To add a vertex

1. If you only wish to work with data on specific layers, turn off any layers that you do not want to modify.
2. Select the Edit|Vertex|Add command. A bounding box is attached to the cursor.
3. Click on the point in the desired line segment where you wish to add a vertex.

   When you select a line segment, any segment that falls within the bounding box is a potential selection. You can decrease the size of the bounding box, using the PgDn key, to avoid selection of more than one segment. If more than one line segment falls within the bounding box, the Choose Selection dialog box appears with a list of segments. Click on the segment you wish to add a vertex to (it is highlighted in the display when you choose it from the list), and click OK.
4. The segment is attached to the cursor. Move the cursor to the location where you wish to place the new vertex, and click to place it.
5. You may continue to add vertices to other line segments, or press the Esc key to exit the function.

Related topics

Add|Vertex

Edit|Vertex|Delete

Toolbar Button: 

This command allows you to delete a vertex anywhere on an existing line segment or mill path.

To delete a vertex

1. Select the Edit|Vertex|Delete command.

⚠ *To avoid the selection of more than one vertex, use the PgDn key to decrease the size of the bounding box until it is a reasonable size for selecting the data in the workspace.*

2. Click on the vertex you wish to delete. Any vertex that falls within the bounding box is a potential selection. If more than one is selected, a Choose Selection dialog box appears with a list of segments. Click on the segment you wish to remove the vertex from (it is highlighted in the display when you choose it from the list), and click OK.
3. You are asked to confirm the affected line segments. Confirm the selection, and the vertex is immediately deleted.
4. You may continue to delete vertices, or press the Esc key to exit the function.

Edit|Vertex|Move

Toolbar Button: 

This command allows you to move an existing vertex.

⚠ *Tip: The current orthogonal line snap setting affects this function (if the Ortho Line Snap is turned on).*

To move a vertex

1. Select the Edit|Vertex|Move command.
2. Click on the vertex you wish to move. The affected line segments are highlighted and attached to the cursor.
3. Move the cursor to the new location, and click to confirm the placement.
4. You may continue to move vertices, or press the Esc key to exit the function.

Edit|Vertex|Segment Delete

Toolbar Button: 

This command allows you to delete a line segment anywhere on an existing line or mill path.

To delete a line segment

1. Select the Edit|Vertex|Segment Delete command.

⚠ *To avoid the selection of more than one segment, use the PgDn key to decrease the size of the bounding box until it is a reasonable size for selecting the data in the workspace.*

2. Click on the segment you wish to delete. Any segment that falls within the bounding box is a potential selection. If more than one is selected, a Choose Selection dialog box appears with a list of affected lines/paths. Click on line/path that contains the segment you wish delete (it is highlighted in the display when you choose it from the list), and click OK.
3. The selected segment is highlighted, and you are asked to confirm the deletion. Confirm the selection, and the segment is immediately deleted.
4. You may continue to delete segments, or press the Esc key to exit the function.

Edit|Clip

Toolbar Button: 

The Clip command erases all data in a selection window. Its main purpose is to allow you to selectively cut data from a draw, mill path, NC circle, etc. based on a window boundary given. If you clip a drilled circle, the data are transformed to individual drill hits. If you clip a milled circle, the data are transformed to a standard mill path.

It can be used to delete all types of data in a selection window. However, if you only wish to delete entire draws, flashes, etc., use the Edit|Delete command.

To clip only drawn data

1. Select the Edit|Clip command.
2. In the Selection Filter, select Window +Xing mode, and the desired data Types. Also specify the layers and D-codes/Tools you wish to work on.

3. Create a selection window around the drawn data you wish to delete. The data are immediately clipped, leaving the rest of the draw intact.
4. When you are finished, press the Esc key to exit the function.

Edit|Join

Toolbar Button: 

The Join command is used to join two line segments or mill paths together.

This function will not join segments within the same mill path. If you wish to do so, select the Edit|NC|Explode command to explode the path segments into individual paths, select the Edit|Join command to join the desired segments, then the Edit|NC|Combine Paths command to combine the paths back into a single path.

To join two lines

1. Select the Edit|Join command. The Join dialog box appears.
2. You can join line segments using one of three Modes.

   Normal Mode
   Using Normal mode, two selected line segments are extended or trimmed as needed so that they connect at a vertex. If you wish to eliminate this vertex later, you must first convert the lines to polylines, using the Tools|Convert|Draws To Polylines command. Then delete the vertex using the Edit|Vertex|Delete command.

   If you are joining two lines that currently intersect, and want to only trim one of the lines to the intersection point, select the Trim Second Line To First Only command. This is useful if you have one long line in with several intersecting lines. Otherwise, both lines will be trimmed to the intersection point. This option is ignored when joining mill paths.

   Note that, with this mode, you can join lines that are in-line with each other, but not parallel lines.

   Chamfer Mode
   Chamfer connects the lines first by extending or trimming them, then a third line is inserted to join them. Use the Size option to specify how far back from the intersection of the two lines the system is to place the chamfer. If the two lines are in-line or parallel to each other, this mode will not work.

   Fillet Mode
   Fillet connects the lines first by extending or trimming them, then an arc is inserted to join them. Use the Size option to specify how far back from the intersection of the two lines the system is to place the fillet. If the two lines are in-line or parallel to each other, this mode will not work.
3. When you are finished making your selections, click OK. You are prompted to select the two lines to join.
4. Using the Selection Filter, if necessary, click on the two lines you wish to join. The lines are joined immediately.
5. You can select other lines to join, or press the Esc key to end the function.

Edit|Rotate

Toolbar Button: 

The Rotate command rotates a windowed selection or Select Group of items, on a user-defined pivot point or an automatically-calculated center of the data.

If you wish to rotate items in a Select Group, you must first create the group using the Edit|Select|New Group command.

To rotate items

1. Select the Edit|Rotate command. The Rotate dialog box appears.
2. The Pivot Point options allow you to select the pivot point for rotating the selected items. If Center is chosen, the center of the selected items is automatically calculated and used. If Interactive is chosen, you are prompted for this point.
3. Select the number of Degrees, counter-clockwise, to rotate the items.
4. If you want D-codes rotated as well, select the Rotate D-codes option. This is useful if you have custom apertures with an irregular shape whose orientation to the other data must be maintained. New D-codes will be created as necessary. If you do not select this option, the D-codes will not be rotated with the data.
5. Click the OK button when you are finished making your selections.
6. Using the Selection Filter, select the items you wish to rotate.
7. After making your selection, if you chose a Center axis point, the data are immediately altered.

-or-

If you chose an Interactive pivot point, you must now select the pivot point. You can do this by clicking on the point in the workspace, or selecting exact coordinates in the XY Bar. Your data are then altered.

8. Select other data to rotate, or press the Esc key to exit the function.

Related topics

Edit|Mirror

Edit|Mirror

Toolbar Button: 

Use this command if you need to mirror (flip) a windowed selection or Select Group of items either horizontally or vertically. This command modifies your data. If you have data that you just wish to view as if looking at it from behind, use the View|Backside command.

If you wish to mirror a Select Group, you must first create one using the Edit|Select|New Group command.

To mirror data

1. Select the Edit|Mirror command. The Mirror dialog box appears.
2. The Pivot Point options allow you to select the axis point for mirroring the selected items. If Center is chosen, the center of the selected items is automatically calculated and used. If Interactive is chosen, you are prompted for this point.
3. Select whether to have the items mirrored horizontally or vertically.
4. If you want D-codes mirrored as well, select the Mirror D-codes option. This is useful if you have custom apertures with an irregular shape whose orientation to the other data must be maintained. New D-codes will be created as necessary. If you do not select this option, the D-codes will not be mirrored with the data.
5. Click the OK button when you are finished making your selections.
6. Using the Selection Filter, select the items you wish to mirror.
7. After making your selection, if you chose a Center axis point, the data are immediately altered.

   -or-

   If you chose an Interactive axis point, you must now select the axis point. You can do this by clicking on the point in the workspace, or selecting exact coordinates in the XY Bar. Your data are then altered.
8. Select other data to mirror, or press the Esc key to exit the function.

Related topics

Edit|Rotate

Edit|Scale

Toolbar Button: 

This command allows you to apply coordinate offsets or scaling to layers. This command is used to compensate for shrinkage during manufacturing.

Warning: This scaling makes no changes to the size of D-codes. To scale D-codes, use Edit|D-Code|Scale.

This function starts at the 0:0 origin and applies the scaling from there. If you wish to apply the scaling from a different start point (such as the center of a panel to compensate for shrinkage), use the Edit|Origin command to change the location of the origin.

If you wish to scale or offset a Select Group of items, you must first create one using the Edit|Select|New Group command.

To scale layers

1. Select the Edit|Scale command. The Scale And/Or Offset Layers dialog box appears.
2. Specify the Scale, which is the scaling factor to apply to the coordinates of an item or layer. For example, if you apply a scaling factor of 2 to the X and Y coordinates of a flash at 4850:5050, the size remains the same, but its new coordinates become 9700:10100. When applied to an entire layer, everything is "expanded" relative to the scale factor. A scale factor of 1 makes no change to the data.
3. Specify an Offset, which adds the specified amount to the coordinates of your selection. For example, an offset value of 1 to the X and Y coordinates of a flash at 4850:5050 moves the flash to 4851:5051. When applied to an entire layer, everything is moved by one X:Y coordinate. An offset value of 0 makes no change to the data.

4. Click OK.
5. Using the Selection Filter, select the data to offset and/or scale. The changes are immediately made in the workspace.
6. Press the Esc key to exit the function.

Edit|D-Code

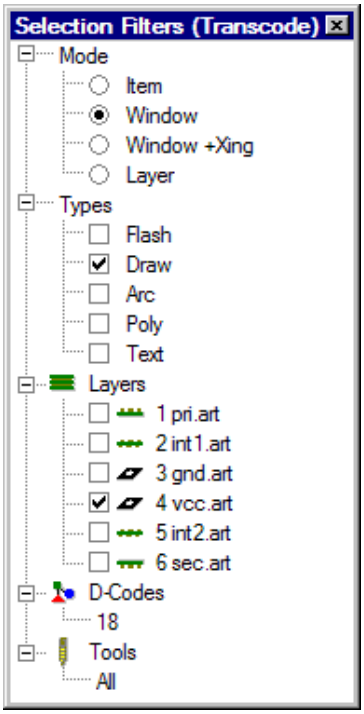
The D-code commands are used to modify the apertures used to create your design.

Edit|D-Code|Transcode

Toolbar Button: 

This command allows you to change the D-code of an individual item, window, group or complete layer. By changing the D-code of an item, you can alter its size and shape. Using the Selection Filter you may specify which items have their D-codes transcoded.

Another way to change an item's size and shape is to edit the aperture list directly.



To change item D-codes

1. Select the Edit|D-Code|Transcode command.
2. Using the Selection Filter, select the item or items whose D-code you wish to change.

   For example, to transcode only draws, using D-code D18, only on layer 4 and only within a particular window, the Selection Filter would need to be set as in the provided illustration.
3. The Transcode dialog box appears. Select the new D-code and click OK

   If you are making several selections, and using the same new D-code for each, you can select the Don't Prompt After Each Selection option in the Transcode dialog box. Then items are assigned the new D-code immediately after selection. If you then need to change the new D-code, you can click the New D-code button in the Status Bar to specify the current replacement D-code.

Edit|D-Code|Explode Customs

Toolbar Button: 

Use this command to "explode" custom apertures in a design. All data within the selected custom apertures are converted into their individual data items such as flashes, drills and draws. This is useful if you have to export your database in the 274-D format, because that format does not support the notion of "custom apertures".

To convert custom apertures into their individual elements

1. Make sure that any layers you wish to work on are visible.
2. Select the Edit|D-Code|Explode Customs command. The Expand Custom D-Codes dialog box appears.
3. In the Layers box, select All to process custom apertures on all visible. To select specific layers to operate on, click the arrow button and select the desired visible layers from the list.
4. In the D-Code box, select All to process all custom apertures. To select specific custom apertures to operate on, click the arrow button and select the desired apertures from the list.
5. Click OK, and the selected custom apertures are processed. A message box appears, telling you how many were affected.

Edit|D-Code|Scale

Toolbar Button: 

Use this command to shrink or expand the size of the D-code used to create one or more selected items in your database. New apertures are automatically added to the aperture list, as needed.

Tip: If the scale command creates a new aperture for you, and you subsequently use the Undo command to undo the scaling, the new aperture will remain in the aperture list. Use the Compact button in the Aperture Table dialog box to remove any unused apertures.

To scale items

1. If you plan to scale any pads, select one of the Tools|Convert|Drawn Pads commands prior to scaling. Scaling drawn pads may result in undesired pad sizes.
2. Select the Edit|D-Code|Scale command. The Scale D-Code dialog box appears.
3. Enter the horizontal and vertical Scale factors.
4. Select Fixed Amount to add the scale values to each item. Otherwise, each item size will be multiplied by the scale values specified.
5. Click OK.
6. Using the Selection Filter as necessary, select one or more items to scale.
7. When you are finished, press the Esc key to exit the function.

Related topics

Edit|Scale

Edit|D-Code|Polarity

Toolbar Button: 

Use this command to control the item-level polarity. When using item level polarity, the ordering of the data is crucial. You may find that you need to move data "in place", thereby placing the "moved" data at the end of the database.

Warning: If you plan to export your data in the 274-D or DXF format, you should not use this function. 274-D and DXF do not support polarity.

To change an item's polarity

1. Select the Edit|D-Code|Polarity command. The Polarity dialog box appears.
2. Select Dark polarity to select positive or viewable items, or Clear polarity for negative items, which clear away positive data in a given area.
3. Click OK.
4. Using the Selection Filter as necessary, select the items to apply the polarity to.
5. When you are finished, press the Esc key to exit the function.

Edit|Align Layers

Toolbar Button: 

This command aligns layers for proper layer registration when multiple layers are viewed simultaneously. Proper layer alignment is also crucial to the successful creation of a netlist.

If you wish to verify that your layers are correctly aligned, you can check for layer-to-layer registration using the Analysis|DRC/MRC command.

To align layers

1. Determine the layer to which all other layers should be aligned with (the master layer).
2. Find an item on the master layer to use as a "base item", or reference point.

Warning: Arcs cannot be used as reference points to align layers.

3. Select the Edit|Align Layers command.
4. Click on the item you chose as a master reference point. Use the Selection Filter, if necessary.
5. Click on an item on each layer to be aligned that corresponds to the reference point. As you select each item, it is highlighted and you are asked to confirm your selection. The entire layer is then aligned with the master layer.

   Again, use of the Selection Filter may be helpful here. You may also use the zoom in/out and pan hotkeys to make it easier to find the master reference and corresponding items.
6. If you wish to select a new base item, press the Esc key and repeat steps 4 and 5.

   -or-

   When you are finished, press the Esc key twice to exit the function.

Edit|Snap Pads

Unlike the Edit|Align Layers command, which shifts the entire layer based on an individual reference point, the Snap Pads command examines each individual item on a layer and aligns it to the nearest item on the "golden" layer. This is useful if you need to cleanup low resolution artwork that does not line up properly from layer to layer.

To snap pads

1. Select the Edit|Snap Pads command. The Snap Pads dialog box appears.
2. Select the Golden Layer whose pad or drill locations will be used as a "master" reference to align items on the Snap Layer(s).
3. Select the Snap Layers that contain pads or drills to be aligned with those on the Golden Layer.
4. In the Tolerance box, enter the distance that coordinates of items on the Snap Layer can deviate from the items on the Golden Layer to be considered for alignment. Items within this distance are aligned, items outside this distance are ignored.
5. Click OK. Any qualified data are aligned and you are informed when the process is complete.

Edit|Origin

Toolbar Button: 

This command allows you to relocate the origin (0:0 point) of the database.

To change the origin

1. Select the Edit|Origin command. The Change Database Origin dialog box appears.
2. If you select Database Updates Now, your new origin selection is immediately committed to the database, and the film box and data in the display are altered to reflect it. If you select Database Updates After Next Reload, the new origin is not committed to the database until after you save the database and reload it. The film box will automatically relocate to the origin you select, but none of the existing data (or imported data) will reflect the new origin until after the reload.
3. Select Display This Dialog Only If Shift Key Is Pressed to cause the dialog box to appear in the future if you press the Shift key at the same time that you select the Edit|Origin command. If you select this option, and setting you select above becomes the default for the Edit|Origin command.
4. Click OK, and you are prompted to select a point to define the new origin.
5. Select the new point, and press the Esc key to exit the function.

Edit|NC

The NC commands are used to modify or delete the NC (Drill and Mill) data in your design.

Edit|NC|Drill Properties

This command allows you to change the tool assigned to one or more drill hits, and whether the hits are plated or unplated.

To change drill properties

1. Select the Edit|NC|Drill Properties command.
2. Using the Selection Filter, select one or more drill hits you wish to change. The Drill Properties dialog box appears.
3. Change the drill properties as desired, and click OK.

Tip: A tool must have a Plated status of "Both" in the NC Tool table in order for you to change the Plated status here.

4. You can select additional drill hits to change, or press the Esc key to exit the function.

Related topics

Edit|NC|Change|Tools

Edit|NC|Path Properties

Toolbar Button: 

This command allows you to change the tool assigned to one or more mill paths, as well as apply compensation to a path.

Most NC tool characteristics are defined in the NC Tool Table. This command allows you to change the feed rate and plated status on an item-by-item basis.

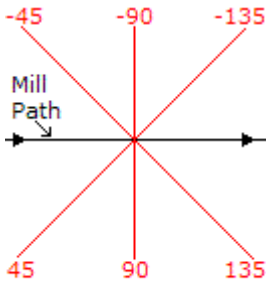
To change a mill path's properties

1. Select the Edit|NC|Path Properties command.
2. Using the Selection Filter, select one or more mill paths you wish to change.
3. If you selected a standard mill path, the Mill Path Properties dialog box appears. Change the path properties as desired.

Tool
The tool number to use for this path.

Plated
Whether the path should be plated or unplated. A tool must have a Plated status of "Both" in the NC Tool table in order for you to change it here.

Plunge
If you have a plunge point that lies outside of the mill path, this shows Angle and Length for the plunge line. The plunge point is based on these settings and the starting point of the path. The Angle is the deviation (counter-clockwise for positive angles, clockwise for negative angles) of the plunge line from the first segment of the mill path, or the tangent at the starting point (if the first segment is an arc). The angle is limited to a range of +/- 135 degrees. Specify a value of "0" for both options, if you want the plunge point to be on the mill path (at the starting point).



Plunge Line Angle

Overshoot
Instructs the tool to cut the specified distance beyond the ends of the mill path before extraction.

Feed Rate
The rate at which the mill tool travels through the material to create the paths, in either inches per minute or mm per second (depending upon your units of measure). The default rate (when this option is set at "Default" or "0") is the feed rate assigned in the NC Tool Table. If you specify a different rate here, then it will override the setting in the Tool Table.

Compensation
A compensated mill path is offset from the centerline by the radius of the mill tool, or the value set in the Compensation Index Table in the NC Tool Table. If you wish to use compensation, specify whether the path should be offset to the Right or Left, relative to the direction that the mill tool travels.

-or-

If you selected a milled circle, the Mill Circle Properties dialog box appears. Change the circle properties as desired.

Tool
The tool number to use for this circle.

Feed Rate
The rate at which the mill tool travels through the material to create the circle, in either inches per minute or mm per second (depending upon your units of measure). The default rate (when this option is set at "Default" or "0") is the feed rate assigned in the NC Tool Table. If you specify a different rate here, then it will override the setting in the Tool Table.

Plated
Whether the circle should be plated or unplated. A tool must have a Plated status of "Both" in the NC Tool table in order for you to change it here.

Direction
Select CW to have the circle routed in a clockwise direction, or select CCW to have the circle routed in a counter-clockwise direction.

Type
Select Inside to have the circle routed on the inside of the specified radius, or select Outside to have it routed on the outside of the specified radius.

4. Click the OK button.
5. You can select additional mill paths to change, or press the Esc key to exit the function.

Related topics

Edit|NC|Change|Tools
Edit|NC|Reverse Path Direction

Edit|NC|Change

The change commands are used to modify any operator messages, NC tools, or break tabs used in your design.

Edit|NC|Change|Operator Message

Toolbar Button: 

Use this command if you want to change the text in an operator message.

To change an operator message

1. Select the Edit|NC|Change|Operator Message command. You are prompted to click on the NC item that has the desired operator message associated with it.
2. Click on the drill or mill item with the operator message. The NC Operator Message dialog box appears.
3. Edit the text, as desired, and click OK.
4. You can select other operator messages to change, or press the Esc key to exit the function.

Related topics

Edit|NC|Delete|Operator Message

Edit|NC|Change|Tools

Toolbar Button: 

This command allows you to change NC tool assignments for existing database items. You may select a single item, or multiple items by window or entire layers.

To change tools

1. Select the Edit|NC|Change|Tools command. You are prompted to select the NC item whose tool you wish to change.
2. Using the Selection Filter, select the NC item or items you wish to change tool assignments for. The Change NC Tool dialog box appears.
3. Select the New Tool that you wish to apply to all the selected data, and click OK. The tools are changed.
4. Select other items to change, or press the Esc key to exit the function.

Related topics

Edit|NC|Drill Properties
Edit|NC|Path Properties

Edit|NC|Change|Break Tab

Toolbar Button: 

If you need to change a mill tab that you have added to a mill path, make sure the new mill tab is defined in the Setup|Break Tabs function, then use the Edit|NC|Change|Break Tab command.

If you wish to change all occurrences of a particular break tab, you can simply redefine the break tab in the Setup|Break Tabs function. When you change a tab's definition there, all occurrences of that tab in your database are changed to reflect the new definition.

To change a break tab

1. Select the Edit|NC|Change|Break Tab command. You are prompted to select the break tab to change.
2. Click on the break tab you wish to change.

Tip: If you are zoomed in fairly close on the tab, you may note the point of selection is fairly limited. On a normal break tab or stitched tab, click on an end point of the mill path. On a crown or crown stitched break tab, click where the mill path bends to create the "legs".

-or-

If you wish to change multiple break tabs to the same tab, click on the Window or Window +Xing mode in the Selection Filter, and create a selection window around the tabs you wish to delete. All the selected tabs are highlighted.

3. A shortcut menu is attached to your cursor, with the names of the defined break tabs. Click on the break tab you wish to change to. The tab is immediately changed.

4. You may change additional tabs, if desired. Press the Esc key when you are finished.

Related topics

Edit|NC|Delete|Break Tab
Edit|NC|Move Break Tab

Edit|NC|Change|Mill Plunge Point

This command allows you to change the location of your plunge point. If you wish to change the length or angle of your plunge point, use the Edit|NC|Path Properties command.

When you select this command, the view of any compensation, plunge line, or overshoot that is applied to the mill path is turned off, as you are modifying the "base" mill path.

To change a plunge point location

1. To view the locations of the vertices in your mill path more easily, it is recommended that you select the View|Sketch command.
2. Select the Edit|NC|Change|Mill Plunge Point command.
3. Select the path whose plunge point you wish to change.
4. Click on the vertex where you wish to place the plunge point. The plunge point is immediately relocated.

Tip: If a vertex does not exist at the location you wish to place the plunge point, use the Add|Vertex command to create one.

5. Press the Esc key to end the function. At this point, if you previously defined any length or angle to your plunge point, the workspace is updated to reflect the definitions.

Edit|NC|Delete

The Delete commands are used to remove NC data from your design.

Edit|NC|Delete|Operator Message

Toolbar Button: 

Use this command to delete an operator that is associated with an NC item.

To delete an operator Message

1. Select the Edit|NC|Delete|Operator Message command. You are prompted to select the NC item that has the operator message associated with it.
2. Click on the desired item, and its associated operator message is deleted.
3. Select additional operator messages to delete, or press the Esc key to exit the function.

Related topics

Edit|NC|Change|Operator Message

Edit|NC|Delete|Optional Stop

Toolbar Button: 

Use this command to delete an optional stop that is associated with an NC item.

To delete an optional stop

1. Select the Edit|NC|Delete|Optional Stop command. You are prompted to select the NC item that has the optional stop associated with it.
2. Click on the desired item, and its associated optional stop is deleted.
3. Select additional optional stops to delete, or press the Esc key to exit the function.

Edit|NC|Delete|Break Tab

Toolbar Button: 

Use this command to delete break tabs from mill paths.

To delete a break tab

1. Select the Edit|NC|Delete Break Tab command. You are prompted to select the break tab to delete.
2. Click on the break tab you wish to delete. It is immediately removed, and the mill path is restored accordingly.

Tip: If you are zoomed in fairly close on the tab, you may note the point of selection is fairly limited. On a normal break tab or stitched tab, click on an end point of the mill path. On a crown or crown stitched break tab, click where the mill path bends to create the "legs".

-or-

If you wish to delete multiple break tabs, click on the Window or Window +Xing mode in the Selection Filter, and create a selection window around the tabs you wish to delete. They are all immediately removed.

3. Delete additional tabs, if desired. Press the Esc key when you are finished.

Related topics

Edit|NC|Move Break Tab
Edit|NC|Change|Break Tab

Edit|NC|Move Break Tab

Toolbar Button: 

Use the Move Break Tab command to move a tab to a different location in a mill path.

To move a break tab

1. Select the Edit|NC|Move Break Tab command. You are prompted to select the break tab.
2. Click on the break tab you wish to move. A box is attached to the cursor, representing the width of the tab. An anchor point is also attached to the mid-point of the current location.

Tip: If you are zoomed in fairly close on the tab, you may note the point of selection is fairly limited. On a normal break tab or stitched tab, click on an end point of the mill path. On a crown or crown stitched break tab, click where the mill path bends to create the "legs".

3. Click on the mid-point of the location you wish to move the tab to. The tab is immediately moved. (You will not be allowed to move a tab to a location where there is insufficient space in the mill path.)
4. Move additional tabs, if desired. Press the Esc key when you are finished.

Related topics

Edit|NC|Change|Break Tab
Edit|NC|Delete|Break Tab

Edit|NC|Reverse Path Direction

Toolbar Button: 

Use this command to reverse the direction a tool travels to create a mill path (as shown by the arrows displayed in the mill paths). After reversing a mill path's direction, any compensation is automatically reapplied in the defined direction (which is always in relation to the direction of the mill tool).

Related topics

Edit|NC|Path Properties

Edit|NC|Explode

Toolbar Button: 

This command explodes selected NC data into their base components. Drilled items such as circles, slots, and text are converted to their individual drill hits. Pattern drills are exploded into their component NC parts. Mill paths are exploded into multiple mill paths, with one segment (line or arc) per path. Milled circles are similarly exploded into multiple mill paths, with one segment per path.

Tip: If you right-click on an NC item, the explode function is included in the shortcut menu, along with other editing functions. This provides a quick way to explode individual items.

To explode NC data

1. Select the Edit|NC|Explode command.
2. Using the Selection Filter, click on the NC data you wish to explode.
3. The data are highlighted, and you are asked to confirm your selection.
4. Click the Yes button, and the data are converted into their individual elements.

Related topics

Edit|NC|Combine Paths

Edit|NC|Combine Paths

This command allows you to select two or more separate mill paths, and join them into one path. It is essentially the opposite of the Edit|NC|Explode command.

This function does not trim, extend, or otherwise alter the path segments. If the paths do not touch, they will not be extended to meet each other, and any existing vertices will remain. If you wish to combine two mill path segments into a single, contiguous path, extending the existing paths to meet each other, if necessary, use the Edit|Join command.

To join mill paths

1. Select the Edit|NC|Combine Paths command.
2. Using the Selection Filter, window around the paths you wish to join.
3. The paths are highlighted, and you are asked to confirm your selection.

   If the paths that you combined used different tools, compensation, etc., the combined path will automatically be changed to uniform properties. If you wish to view or change the path properties, select the Edit|NC|Path Properties command.

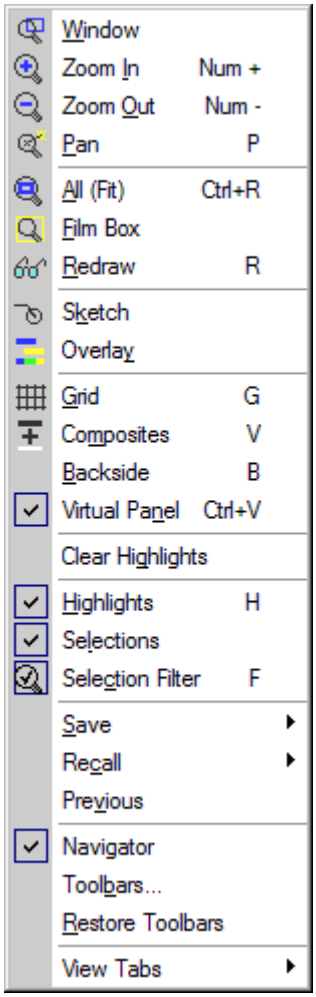
Edit|Purge

Use this command to compact the currently loaded database for more efficient use of memory. Since VisualCAM does not actually erase data from memory during edits, memory may become fragmented and less efficient. Therefore, occasional purging may help VisualCAM perform optimally.

Warning: Purging destroys any Undo information that currently exists. Do not use this command unless you are sure you do not need to undo any previous edits!

View menu

The View menu commands control the viewing window location and size, as well as the type of data you wish to look at.



View|Window

Toolbar Button: 

Use the View|Window command when you want precise control over the view in the workspace. Two points are required to define a rectangle that encompasses the area that is to become the new viewing window.

If you are currently zoomed in very close to the workspace, and wish to expand your view, use the View|Zoom Out command.

To create a view window

1. Click on the upper-left corner of your desired view space. A window is attached to the mouse cursor.
2. Move the cursor and click on the lower-right corner of your desired view space. The workspace immediately zooms in on your selection.

Related topics

View|Zoom In
View|Pan

View|Zoom In

Toolbar Button: 

Equivalent Hotkey: I or **+ (plus key)**

Use the Zoom In command to quickly decrease the size of the viewing window, and see a smaller area of your design in more detail, in the workspace.

If you would like more precise control over the area you wish to view, use the View|Window command.

To zoom in

1. Select the View|Zoom In command. You are prompted to click on a center point.
2. Click on the point in the workspace that you wish to be the center point of your viewing window. The size of the current viewing window is reduced by half.

View|Zoom Out

Toolbar Button: 

Equivalent Hotkey: O or **– (minus key)**

Use the Zoom Out command to quickly increase the size of the viewing window, and see more of your design in the workspace.

To zoom out

1. Select the View|Zoom Out command. You are prompted to click on a center point.
2. Click on the point in the workspace that you wish to be the center point of your viewing window. The size of the current viewing window expands (doubles in size).

View|Pan

Toolbar Button: 

Equivalent Hotkey: P

The Pan command moves the current viewing window to a new location, which is centered on a point you specify. This command does not change the size of the viewing window.

⚠ *Tip: Use Ctrl+P to toggle "Autopan" mode. In this mode, when you move your cursor at the edge of the workspace, the workspace view automatically pans in that direction.*

To pan your view

1. Select the View|Pan command. You are prompted to select a new center point for your view.
2. Click on the new view window center point in the workspace.

   -or-

   Click the Abs button in the XY Bar and select the exact coordinates for the center of your new view window.

   -or-

   Click the Rel button in the XY Bar and select coordinates that are relative to the current center point of the view window.

   After you select the center point, the display immediately pans to the new location.

Related topics

View|Window

View|All (Fit)

Toolbar Button: 

Equivalent Hotkey: Ctrl+R

This command adjusts the size of the viewing window to encompass the extents of the currently displayed layer(s).

⚠ *Tip: If the extents aren't correctly displayed, data has been added to or deleted from the displayed layers since opening the database. Use the Query|Extents command to have the system recalculate the current extremes of the database.*

View|Film Box

Toolbar Button: 

The View|Film Box command adjusts the size of the viewing window to display the extents of the Film Box. To control the color of the film box, select the Options|Configure command, and click the Display tab.

⚠ *Tip: If you want to hide the film box from view, change its color to the same color as the background of the workspace (usually black).*

To view all the items in your database, use the View|All (Fit) command.

View|Redraw

Toolbar Button: 

Equivalent Hotkey: R

The Redraw command refreshes the display in the workspace.

View|Sketch

Toolbar Button: 

The View|Sketch command toggles sketch mode on/off. When sketch mode is enabled, items are shown with an outline only.

An option to this setting is stick mode, which is enabled using the Options|Configure command. (Select the Display Draws As Sticks In Sketch Mode option under the Display tab). When enabled, stick mode displays all draws as a single thin line. This mode can help you spot stacked and buried items.

Related topics

View|Overlay

View|Overlay

Toolbar Button: 

This command toggles overlay mode on/off. When overlay mode is enabled, items on layers become transparent when drawn over those other layers. When overlay mode is disabled, items on layers obscure whatever is drawn underneath them. Overlay mode makes it easier to spot stacked and buried items. Flashes also display significantly faster in overlay mode.

Related topics

View|Sketch

View|Grid

Toolbar Button: 

Equivalent Hotkey: G

The View|Grid command toggles the grid display on/off.

Use the Options|Configure command (select the Display tab) to control the size of the grid.

Related topics

Options|Grid Snap

View|Composites

Toolbar Button: 

Equivalent Hotkey: V

The View|Composites command toggles the way composite layers are displayed, and also toggles the view of item-level polarity. When this command is enabled, the polarity of each item and composite layer—specified using Edit|Item and Setup|Composites, respectively—will be honored; if a layer/item is specified as "Clear," it will be displayed in the background color. When this command is not selected, all items appear as positive (visible) data, regardless of their actual polarity.

Warning: When View Composites is enabled, the active layer is not displayed on top, as it normally is.

View|Backside

Equivalent Hotkey: B

This command essentially "flips" all your data on the Y axis so that you are looking at the design from the back side. This flips your grid as well, so none of the coordinates of your data are changed. This command is a toggle, so you may select it again to view your data from the front.

This command does not physically alter your design. If you wish to mirror your data, select the Edit|Mirror command.

View|Virtual Panel

Equivalent Hotkey: Ctrl+V

The display of virtual panels (created using the Tools|Panelize|Simple command) may be toggled on/off using the View|Virtual Panel command.

View|Parts

If you have parts defined in your database, the data (such as pin numbers, centroids, etc.) exist as "virtual entities". This command is used to toggle the visibility of part data on and off.

View|Clear Highlights

The Clear Highlights command removes all highlights currently displayed in the workspace, such as those that appear when you use the Query|Item command. It does not affect your design data.

Related topics

View|Highlights

View|Highlights

Equivalent Hotkey: H

This command toggles the display of normal highlights on/off. Normal highlights are all highlights not indicating a select group or DRC error.

Related topics

View|Clear Highlights

View|Selections

The View|Selections command toggles the highlighting of the current select group (if any) on/off. This does not affect any of your data -- just its display.

View|Selection Filter

Toolbar Button: 

Equivalent Hotkey: F

The View|Selection Filter command toggles the display of the Selection Filter box on/off. You can also use the "X" button in the upper-right corner of the Selection Filter to close it.

You can only use the selection filters when you are in an applicable function, such as Edit|Delete or Query|Item. Otherwise, you will only see the various filter categories ("Mode", "Types", etc.) without any available selections.

Related topics

Selection Filter

View|Save

The View|Save command is used to save the current viewing window for later recall. You can save up to eight views.

To save a view window

1. Zoom/pan to the view you wish to save in the workspace.
2. Select View|Save and the number you want associated with that view.

Use the View|Recall command to recall any of the saved viewing windows.

Related topics

View|View Tabs|Add
View|Previous

View|Recall

The Recall commands are used to display previously saved viewing windows, which you defined using the View|Save command. Only those positions that have views saved to them are available for selection.

View|Previous

The View|Previous command is used to recall the last viewing window. This allows you to quickly toggle your view between two locations in the workspace.

Related topics

View|Save
View|View Tabs|Add

View|Navigator

This toggle command controls the display of the Navigator. Select View|Navigator to display it, unselect the command to hide the Navigator.

View|Toolbars

Each button within the toolbar represents a shortcut method of launching a menu command. When you click on a button in the toolbar, the command associated with that button is executed. An example of a toolbar with commonly-used functions, such as File|New and File|Save, appears below.



For an explanation of each toolbar, see *Toolbar Button Reference*.

All toolbars and control bars may be moved to a location you prefer by clicking your mouse on the bar and dragging the bar to a new location. They may be docked to an edge of the VisualCAM window, or may float in a small window. You can control which toolbars appear in the window with the View|Toolbars command. You can change which button appears in each toolbar by using the Options|Customize Toolbar command.

To control which toolbars appear

1. Select the View|Toolbars command. The Visible Toolbars dialog box appears, with a list of all available toolbars.
2. Place a check next to each toolbar you wish to view, or remove the check for those you wish to hide, by clicking on your desired selection. Click the All On button to view all toolbars.
3. When you are finished, click the OK button.

Related topics

View|Restore Toolbars

View|Restore Toolbars

The Restore Toolbars command returns the toolbars, Navigator, Selection Filter, Layer Bar, and Item Properties Display to the program default visibility and locations.

Related topics

View|Toolbars

View|View Tabs

VisualCAM allows you to maintain different view windows of your workspace, so that you are not required to frequently zoom and pan your view. You can perform edits, such as copy data, from one view to another. The View Tabs, at the bottom of the workspace, are used to select which view you want at any given time. By default, two views are provided:

- **Log** shows a record of all your actions since you opened or created a database
- **Main** shows the graphical display of your database

You can add additional tabs as necessary, and later delete them.

View|View Tabs|Add

VisualCAM allows you to maintain different view windows of your workspace. The View Tabs, at the bottom of the workspace, are used to select which view you want at any given time. By default, two views are provided:

- **Log** shows a record of all your actions since you opened or created a database
- **Main** shows the graphical display of your database

You can add additional View Tabs, maintaining one view of your database in each, so that you are not required to frequently zoom and pan your view. You can perform edits, such as copy data, from one view to another.

To add a view

1. Select the View|View Tabs|Add command.
2. You are prompted to assign a name to the new tab. You can create your own, or accept the default name assignment. The new tab is created immediately, and your database is fitted to the display.

Related topics

View|Save
View|Previous

View|View Tabs|Delete

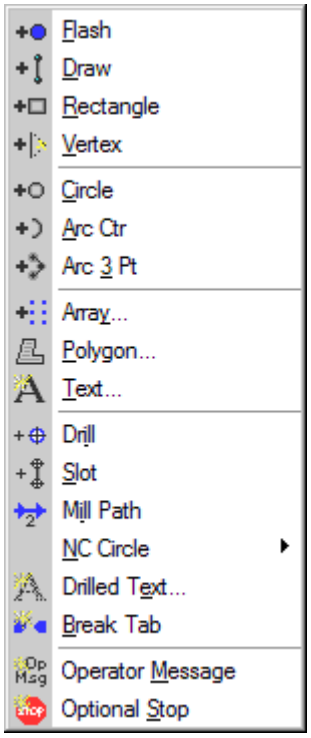
If you have added additional views to the workspace (using the View Tabs|Add command), you can delete them as well. This does not affect any of the design data. You cannot delete the Log and Main view tabs.

To delete view tabs

1. Click on the View Tab you wish to delete.
2. Select the View|View Tabs|Delete command.
3. You are asked to confirm your selection. When you click Yes, the view tab is immediately deleted.

Add Menu

The Add menu commands allow you to enter various types of new database items.



All circles and arcs are created using circular interpolation or with multiple line segments depending on the style indicated by the Options|Arcs 360 command. Use circular interpolation with care as not all photo-plotters support circular interpolation. Segmented circles and arcs use the chord angle specified using the Options|Configure command.

Add|Flash

Toolbar Button: 

This command allows you to add a flash to the active layer.

To add a flash

1. After you select the active layer and the D-code in the Layer Bar, select the Add|Flash command. You are prompted for a point at which to add the flash.

   If the active layer is an NC layer, this becomes the Add|Drill command.
2. As you move the cursor in the workspace, an outline shape of the current D-code is displayed. Click to add a flash at a desired location.
3. You may click on additional locations to add a flash, change the active layer or active D-code, as desired.
4. When you are finished, press the Esc key.

Add|Draw

Toolbar Button: 

This command allows you to draw line segments on the active layer, using the current D-code.

Tip: To add a mill path, use the Add|Mill Path command.

To add a draw

1. After you select the active layer and the D-code in the Layer Bar, select the Add|Draw command.
2. Click on the starting point and subsequent points that form the desired line segments.

   To add arcs, press the A key. This allows you to enter a 3 point arc. After entering the end point of a 3 point arc you may press the 9 key for an automatic 90 degree arc.

   To add a tangential arc (where the start and center points of the arc are tangent to each other), press the A key and then the T key. Then select the end point to the arc.
3. Press the Esc key to end the draw.

4. You may start a new draw, if desired, on the same or different layer with the same or different D-code. Otherwise, press the Esc key to end the function.

Add|Rectangle

Toolbar Button: 

The Add|Rectangle command allows you to draw line segments in the shape of a rectangle. If you add a rectangle to an NC layer, you create a rectangular mill path.

To draw a rectangle

1. In the Layer bar, make the layer you wish to draw the rectangle on the active layer, and select the D-code or Tool you wish to use to create it.
2. Select the Add|Rectangle command. You are prompted to choose the first point.
3. Click on the location of the lower-left corner of your rectangle. The shape of the rectangle is now attached to your cursor.
4. Click on the upper-right corner of your rectangle, to complete it.
5. You may add additional rectangles, or press the Esc key to exit the function.

Add|Vertex

Toolbar Button: 

The Add|Vertex command allows you to add a vertex anywhere on an existing line segment or mill path.

Tip: The current orthogonal line snap setting affects this function (if the Ortho Line Snap is turned on).

To add a vertex

1. If you only wish to work with data on specific layers, turn off any layers that you do not want to modify.
2. Select the Add|Vertex command. A bounding box is attached to the cursor.
3. Click on the point in the desired line segment where you wish to add a vertex.

   When you select a line segment, any segment that falls within the bounding box is a potential selection. You can decrease the size of the bounding box, using the PgDn key, to avoid selection of more than one segment. If more than one line segment falls within the bounding box, the Choose Selection dialog box appears with a list of segments. Click on the segment you wish to add a vertex to (it is highlighted in the display when you choose it from the list), and click OK.
4. The segment is attached to the cursor. Move the cursor to the location where you wish to place the new vertex, and click to place it.
5. You may continue to add vertices to other line segments, or press the Esc key to exit the function.

Related topics

Edit|Vertex|Add

Add|Circle

Toolbar Button: 

This command allows you to add a circle to the active layer, using the current D-code. If the active layer is an NC layer, the current NC tool is used.

All circles are created using circular interpolation or with multiple line segments depending on the style indicated by the Options|Arcs 360 command. Use circular interpolation with care as not all photo-plotters support circular interpolation. Segmented circles and arcs use the chord angle specified using the Options|Configure command.

To add a circle

1. After you select the active layer and the D-code in the Layer Bar, select the Add|Circle command. If the current layer is an NC layer, see the special instructions below.
2. Click on the circle's center point.
3. Move the cursor and click on a point on the radius.
4. Repeat steps 2 and 3 to create another circle. You may select a different layer and D-code to create the circle, if desired. Press the Esc key to exit the function.

To add a circle to an NC layer

If you wish to add a circle to an NC layer, you can also use one of the Add|NC Circle commands, which provide a quicker method of creating your desired circle type. Otherwise, follow the steps below.

1. After selecting the active NC layer and tool in the Layer Bar, select the Add|Circle command.
2. Click on the circle's center point.
3. Move the cursor and click on a point on the radius. The NC Circle dialog box appears.
4. Select the circle type that is appropriate for the active NC tool. If you are creating a milled circle, select whether you want it milled in a Clockwise or Counter-Clockwise direction, and either Inside or Outside of the radius.
5. If you do not want this dialog box to appear the next time you add an NC circle, select the Display This Dialog Only If Shift Key Is Pressed option. The circle definitions you make now will be repeated in the future, and the dialog box will only appear if you press the Shift key when you select the Add|Circle command.
6. Click OK.

Warning: If you select a circle type that does not reflect the current tool type, you will be warned that the tool type will have to be changed. Click OK in the message box to change the tool type, or cancel to abort the entire function.

Add|Arc Ctr

Toolbar Button: 

The arc is drawn on the active layer, using the current D-code, in a counter-clockwise direction. The arc is created as either a 360° interpolated circle or with multiple line segments depending on the settings of the Options|Arcs 360 command. Use circular interpolation with care as not all photo-plotters support circular interpolation. Segmented arcs use the chord angle specified using the Options|Configure command.

Tip: To add an arc to a Mill path, use the Add|Mill Path command.

To add an arc

1. After you select the active layer and the D-code in the Layer Bar, select the Add|Arc Ctr command.
2. Click on the center point of the arc. A circle representing the radius of the arc is attached to the cursor.
3. Click on the point defining the radius and starting point, then click on the end point.
4. Repeat steps 2 and 3 to create another arc. You may select a different layer and D-code to create the arc, if desired. Press the Esc key to exit the function.

Add|Arc 3 Pt

Toolbar Button: 

The arc is drawn on the active layer, using the current D-code. The arc is created as either a 360° interpolated circle or with multiple line segments depending on the settings of the Options|Arcs 360 command. Use circular interpolation with care as not all photo-plotters support circular interpolation. Segmented arcs use the chord angle specified using the Options|Configure command.

Tip: To add an arc to a Mill path, use the Add|Mill Path command.

To add an arc

1. Select the active layer and the D-code/tool in the Layer Bar
2. Select the Add|Arc 3 Pt command.
3. Define an arc by clicking on its end points and then a point on its circumference.

   -or-

   After entering the end points of an arc you may press the 9 key for an automatic 90 degree arc.
4. Repeat step 2 to create another arc. You may select a different layer and D-code to create the arc, if desired. Press the Esc key to exit the function.

Add|Array

Toolbar Button: 

Use this command to add a pattern of D-codes to the active layer. If the active layer is an NC layer, the array contains drill hits using the currently selected tool.

To add an array

1. After you select the active layer and the D-code/tool in the Layer Bar, select the Add|Array command. The Add Array dialog box appears.
2. Define the spacing between items in the pattern, and click OK.
3. Click on the upper-left corner of the rectangular area that will contain the array, and then the lower-right corner. As you move the cursor to the second corner, the current number of rows and columns, along with the total number of items in the array, is displayed in the Status Bar.
4. Repeat steps 2 and 3 to add another array with the same spacing, or press the Esc key to end the function.

Add|Polygon

Toolbar Button: 

This command is commonly used to create ground plane areas. You can create a polygon either by drawing the outline of a new one, or selecting an existing outline. The interior of the polygon is filled using either a raster fill or vector fill method.

Warning: If you plan to export your files in 274-D format, use vector filling. Raster filling is not supported in 274-D.

To create a polygon

1. Select the Add|Polygon command. The Add Polygon dialog box appears.
2. The polygon must be an enclosed area. For the Polygon Border, select Draw if you wish to draw the outline of the polygon. Choose Select to select a pre-existing closed polygon to fill.
3. The results of running the command will be placed onto the selected Destination Layer. Selecting Same causes the polygon to be placed on the active layer. Using the Same option allows you to add polygons on different layers by simply changing the active layer. If you select a specific destination layer, you must restart the Add Polygon function if you wish to add another polygon to a different layer. The active layer is always the source layer for either the data to be selected/created for the border, or the data that are poured around/flooded. If the destination layer is the same as the source layer, and the polygon border is selected, the data making up the border will be deleted.
4. Select a Fill Method.

   In Flood Fill mode, any objects inside the polygon will be covered over.

   In Pour Around mode, the interior of the polygon is filled while maintaining the specified clearances around all objects inside the polygon. Pour Around also recognizes polygon voids, and will pour around them. Draw Clearance specifies the distance the fill is to be kept from the draws in the design. Flash Clearance specifies the distance the fill is to be kept from the flashes in the design.
5. Select either a Raster or Vector fill for the Fill Type. Vector filled polygons are composed of numerous draws, while Raster polygons use special commands inside the Gerber file to perform the same thing.

   Options for Raster Fill
   When you select to Pour Around with a Raster Fill, many smaller polygons may be generated when pouring around the circuitry. The Minimum Size parameter allows you to specify the minimum size polygon area that will be allowed. Each polygon is undersized by half the minimum size; if it implodes, it is deleted.

   Smooth Polygons uses the same minimum size amount to perform a copper sliver analysis of the resulting polygon pour. If any part of the polygon is found to create a potential sliver, it is automatically fixed by smoothing that area of the polygon.

   Options for Vector Fill
   There are four types of fill styles that can be used for the pour. For None, Solid, and Hatched, the Border D-code is the D-code that will be used to draw the border of the polygon.

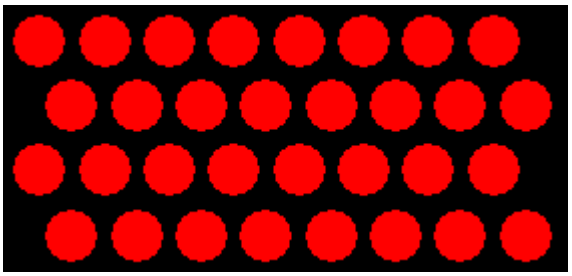
   None creates a polygon outline with no fill.

   Solid creates a polygon with a solid fill. The Fill Dcode is the D-code that is used to create a solid polygon fill. Horizontal specifies that horizontal fill lines are generated, Inset specifies that the edge of the polygon is inset with the width of the Fill D-code until the entire area is filled.

   Hatched creates a hatched polygon fill pattern. Select (check) the number of lines to use in the pattern, then specify the D-code to be used for each line. Step Size values are used to specify the line to line spacing. Angle values are used to specify the angle of the lines used.

   Dotted creates a dotted polygon fill pattern. Select the D-code to use for the dots (flashes), and the spacing between the dots in each row (the X Spacing) and column (the Y Spacing). Specify whether the spacing should be measured from Center-to-Center or Edge-to-Edge. To create a staggered pattern of dots, select the Row Stagger

option and decide which row should be offset. To have them staggered like the below example, the Row 2 offset should be half the value of the Y Spacing amount, and the Row 1 offset should be zero.



6. When you are finished defining the polygon, click OK.
7. If you chose to draw the polygon, click on the various points that make up the polygon. To close the polygon, enter a point at the location where you began the polygon, or automatically close the polygon by pressing the End key.

   -or-

   If you chose to select the polygon border, you are prompted to select a closed polygon to fill, and the Enable/Disable Outset button appears in the Status bar. When Outset is enabled, the resulting filled polygon will be as large as the source line segments, including the entire width of the aperture used to draw it. When Outset is disabled, the resulting polygon will only be as large as the center line of the source line segments.

   Select the desired polygon. If Outset enabled and you use a raster fill, the fill will exactly follow the polygon's border except for the affects of smoothing (if smoothing is selected). If you use a vector fill, the outer edge of the vector polygon's border will follow the outer edge of the selected border, except in interior corners (corners less than 180 degrees) where it will vary from the outer edge if the D-code used to draw the selected border is different from the D-code used for the vector polygon's border. If the selected border is drawn with D-codes of different sizes, there will be discrepancies between the vector polygon's border and the segments of the border drawn with larger D-codes.

Add|Text

Toolbar Button: 

The Add|Text command provides the ability to insert text into the database. Text may be rotated, mirrored or slanted if desired. The height and width of the text is also user-specified as is the character and line spacing. If you wish to change the text after you have added it, use the Edit|Text command.

To add text

1. Make the layer that you want the text associated with the active layer.
2. Select the Add|Text command. If the active layer is an NC layer, you are automatically switched to the Add|Drilled Text command. For other layers, the Text dialog box appears, with the following options:

   Height
   The character height of the text to be added.

   Width
   The character width of the text to be added.

   Rotation
   The angle, in degrees, of each line of text, where 0 is horizontal and 90 is vertical.

   Slant
   The angle of the individual characters in the added text. This can be used to produce italic text and other special effects.

   Line Spacing
   The spacing between lines of text. The value used is a percentage of the actual character height. 1.0 will produce spacing the same height as the characters, 0.5 one half, etc.

   Char Spacing
   The inter-character spacing of the text to be added. 1.0 will produce spacing the same height as the characters, 0.5 one half, etc.

   Mirror
   Selecting this causes the text produced by this command to be mirrored and printed right-to-left, so it can be placed on mirrored Gerber files.

Font
You may select any font from the Font dropdown list. This list contains all the TrueType fonts on your system, and the special "GerbTool-Stroke" font. GerbTool-Stroke is a simple font that does not use polygonal data or negative polarity.

The following is a list of important points to remember when using TrueType fonts:

- TrueType fonts require the use of polygonal data and a combination of positive and negative polarity. When exporting your data, note that, by nature, DXF does not support negative polarity, and the 274-D file format does not support negative polarity or polygonal data.
- VisualCAM modifies the layer setup by adding additional composite layers. Any previously generated report files that specify layer numbers will be subsequently out of sync.
- As TrueType fonts require the use of composites layers, composite viewing must be enabled to view the text properly. While running this command enables this mode automatically, you may use the View|Composites command to toggle this mode on and off.

Text File
To load a file that contains text you wish to insert, click the **Load** button and select the file. This is useful if you have frequently used text.

Clicking the **Save** button saves the contents of the text window to the file named in the Text File box.

Text Entry Field
This area is used to display the text that is to be added. Text is entered into this field by either typing it in, or loading it from a file using the Load button above.

3. When you are finished defining your text, click OK. Your text is now attached to your cursor.
4. Place the text by clicking on the desired location in the workspace.
5. You can place copies of the text in additional locations, by clicking on them, or press the Esc key to end the function.

Add|Drill

Toolbar Button: 

This command allows you to add a drill hit to the active NC layer.

To add a drill

1. Select the desired NC layer and tool from the Layer Bar.
2. Select the Add|Drill command. You are prompted for a location at which to add the drill.
3. Click the mouse on the desired location, or use absolute/relative coordinates from the XY Toolbar.
4. Repeat step 3 for as many drill hits as desired. You can change to different tools and NC layers at any point.
5. When you are finished, press the Esc key to end the function.

Add|Slot

Toolbar Button: 

This command allows you to draw drill slots on the active NC layer.

To add a slot

1. Select the desired NC layer and tool from the Layer Bar. You can only use tools whose Type is defined as "Drill" or "Both".
2. Select the Add|Slot command. You are prompted for a starting location for the slot.
3. Click the mouse on the desired location, and click on the subsequent points that form the slot.
4. After selecting the end point for the slot, press the Esc key to end the slot.
5. You may add another slot by clicking on a new starting point (you can change to different tools and NC layers at any point), or press Esc again to exit the function.

Add|Mill Path

Toolbar Button: 

This command allows you to add a new mill path to an NC layer. If you want to convert drawn lines to a mill path, use the Tools|NC|Create Path From command.

To add a mill path

1. Select the desired NC layer and tool from the Layer Bar.
2. Select the Add|Mill Path command.
3. Click on the starting point and subsequent points that form the desired mill path.

   To add arcs, press the A key. This allows you to enter a 3 point arc. After entering the end point of a 3 point arc you may press the 9 key for an automatic 90 degree arc.

   To add a tangential arc (where the start and center points of the arc are tangent to each other), press the A key and then the T key. Then select the end point to the arc.
4. Press the Esc key to end the path.
5. You may start a new path, if desired, on the same or different layer with the same or different tool. Otherwise, press the Esc key to end the function.

   Note the following symbols are used to represent the plunge and extraction points.



Related topics

Setup|NC Tools
Edit|NC|Path Properties

Add|NC Circle

The NC Circle commands allow you to add circles to NC layers using either drill or mill tools.

Add|NC Circle|Drilled

Toolbar Button: 

This command creates a circle on an NC layer using a drill tool.

To add a drilled circle

1. Select the desired NC layer and drill tool from the Layer Bar.
2. Select the Add|NC Circle|Drilled command. You are prompted to select the circle's center point.
3. Click on the circle's center point.
4. Move the cursor and click on a point on the radius.
5. Repeat steps 3 and 4, as desired to create additional circles. You can also select different NC layers and tools to create those circles.
6. Press the Esc key to exit the function.

Add|NC Circle|Milled

You can add milled circles that are cut on either the inside or outside edge of the specified radius, and have it cut in either a clockwise or counter-clockwise direction.

Add|NC Circle|Milled|CCW Inside

Toolbar Button: 

This command creates a milled circle that is routed on the inside of the radius you specify, in a counter-clockwise direction.

To create a milled circle

1. Select the desired NC layer and tool from the Layer Bar.
2. Select the Add|NC Circle|Milled|CCW Inside command.
3. Click on the circle's center point.
4. Move the cursor and click on a point on the radius.
5. Repeat steps 3 and 4, as desired to create additional circles. You can also select different NC layers and tools to create those circles.
6. Press the Esc key to exit the function.

Add|NC Circle|Milled|CCW Outside

Toolbar Button: 

This command creates a milled circle that is routed on the outside of the radius you specify, in a counter-clockwise direction.

To create a milled circle

1. Select the desired NC layer and tool from the Layer Bar.
2. Select the Add|NC Circle|Milled|CCW Outside command.
3. Click on the circle's center point.
4. Move the cursor and click on a point on the radius.
5. Repeat steps 3 and 4, as desired to create additional circles. You can also select different NC layers and tools to create those circles.
6. Press the Esc key to exit the function.

Add|NC Circle|Milled|CW Inside

Toolbar Button: 

This command creates a milled circle that is routed on the inside of the radius you specify, in a clockwise direction.

To create a milled circle

1. Select the desired NC layer and tool from the Layer Bar.
2. Select the Add|NC Circle|Milled|CW Inside command.
3. Click on the circle's center point.
4. Move the cursor and click on a point on the radius.
5. Repeat steps 3 and 4, as desired to create additional circles. You can also select different NC layers and tools to create those circles.
6. Press the Esc key to exit the function.

Add|NC Circle|Milled|CW Outside

Toolbar Button: 

This command creates a milled circle that is routed on the outside of the radius you specify, in a clockwise direction.

To create a milled circle

1. Select the desired NC layer and tool from the Layer Bar.
2. Select the Add|NC Circle|Milled|CW Outside command.
3. Click on the circle's center point.
4. Move the cursor and click on a point on the radius.
5. Repeat steps 3 and 4, as desired to create additional circles. You can also select different NC layers and tools to create those circles.
6. Press the Esc key to exit the function.

Add|Drilled Text

Toolbar Button: 

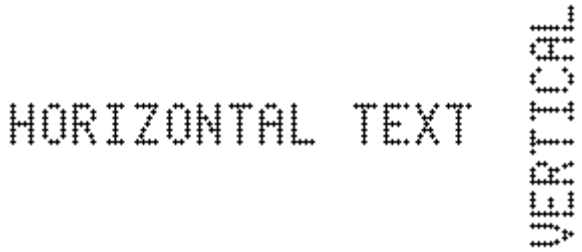
Text that is created with numerous drill hits can be added to any NC layer. The color of the text in the workspace depends upon what colors you have selected for your Drill Display Settings. See the Options|Configure command for more details.

The size of the individual characters in the text depends upon which format you are working with (Excellon or Sieb & Meyer, as specified in the NC Configuration Settings) and the drill tool you use to create it (larger tools will result in larger text).

Because placement of the characters is based on a grid, characters may not all be the size, but the space occupied by each is the same: For Excellon, the size is 4 dill hits wide by 7 hits high, for Sieb & Meyer, they are 5 hits wide by 7 hits high. Both formats define the spacing between the drill hits as 1.2 times the drill size, or 0.0417 inches if no tool size is defined.

To add drilled text

1. Select the desired NC layer as the active layer in the Layer Bar.
2. Select the drill tool you wish to use to create the text as the active tool.
3. Select the Add|Drilled Text command. The Add Drilled Text dialog box appears.
4. Choose whether you want the orientation of the text to be Horizontal or Vertical.



5. The Text Entry field, at the bottom, displays the text that is to be added. Type in your desired text here.

   -or-

   To load a file that contains text you wish to insert, click the Load button and select the file.

   The two drill formats support slightly different character sets, so if a character is not supported, the space is left blank when you place the text in the workspace. Both formats support all alphabetic (upper case only) and numeric characters, and the following special characters:

   Excellon: **+ = / ***

   Sieb & Meyer: + - / ! " () . > ? :
6. If you wish to save the contents of the Text Entry field to a file, enter a file name in the Text File box, and click the Save button. This is useful if you have frequently used text.
7. When you are finished, click the OK button. An outline of the text is attached to your cursor. Click on the location where you want to place the text.
8. You can click on additional locations to place the same text, or press the Esc key to exit the function.

Related topics

Edit|Text

Add|Break Tab

Toolbar Button: 

Use this command to insert tabs into mill paths.

To add a break tab

1. Define your break tabs using the Setup|Break Tabs command.
2. Select the Add|Break Tab command.
3. A list of tab definitions appears in the Layer Bar. Select the tab you wish to insert. A bounding box is added to your cursor, representing the width of the break tab.
4. Click on the point in the desired mill path where you wish to insert the tab.

   You will not be allowed to add a tab to a location where there is insufficient space. The bounding box gives you a nominal size, but the true size factors-in the size of the tool used for the mill path, as well as whether the path is compensated; this cannot be determined until you select the mill path you wish to add the tab to. If the selected point is too close to another tab, or either end of the path, a tab will not be added. *The actual minimum required distance is the edge of the bounding box plus half of the selected path's tool size.* If the amount of space available is between the minimum required and the amount required to insure the tab will fit even if the path is compensated, you will be warned.
5. Add additional tabs, if desired. Press the Esc key when you are finished.

Related topics

Edit|NC|Change|Break Tab
Edit|NC|Move Break Tab
Edit|NC|Delete|Break Tab

Add|Operator Message

Toolbar Button: 

Use this function to insert a command into a drill or mill sequence that causes the machine to stop, either before or after the NC item is created, and display a message to the operator.

Tip: Use the Query|Item command to view any operator messages already associated with an NC item.

To add an operator message

1. Select the Add|Operator Message command. You are prompted to select the NC data to add the message to.
2. Click on the drill or mill data in the workspace. The NC Operator Message dialog box appears.
3. Type your message in the text box, and click OK. The Add NC Operator Message dialog box appears.
4. Click the Before button if you want the message to appear to the operator before the item is created.

   -or-

   Click the After button if you want the message to appear to the operator after the item is created.

   A small blue box appears in the workspace, with an arrow pointing to the NC item that the operator message is associated with.
5. Select another NC item to add an operator message to, or click the Esc key to exit the function.

Related topics

Edit|NC|Change|Operator Message
Edit|NC|Delete|Operator Message
Add|Optional Stop

Add|Optional Stop

Toolbar Button: 

Use this function to insert a command into a drill or mill sequence that causes the machine to stop, either before or after the NC item is created, and extract the tool.

Tip: To see if an NC item already has an optional stop associated with it, use the Query|Item command.

To add an operator message

1. Select the Add|Optional Stop command. You are prompted to select the NC data to add the stop to.
2. Click on the drill or mill data in the workspace. The Add NC Optional Stop dialog box appears.
3. Click the Before button if you want the stop to occur before the item is created.

   -or-

   Click the After button if you want the stop to occur after the item is created.

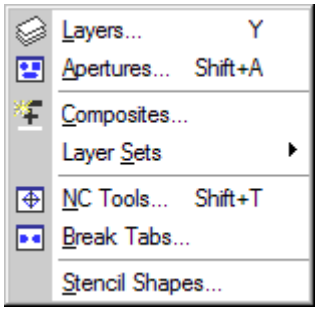
   A small red octagon appears in the workspace, with an arrow pointing to the NC item that the stop is associated with.
4. Select another NC item to add a stop to, or click the Esc key to exit the function.

Related topics

Edit|NC|Delete|Optional Stop
Add|Operator Message

Setup Menu

The Setup menu commands allow you to manage the different tables that VisualCAM utilizes.



Related topics

The Navigator

Setup|Layers

Toolbar Button: 

Equivalent Hotkey: Y

The Setup|Layers command displays the layer table. You can add, edit, delete, and control the visibility of any layer in your database from the layer table. Any changes that you make here will be reflected in the Color Bar and the Navigator once you click OK to close the dialog box.

"Tagging", or identifying, layers is necessary for many VisualCAM utilities to function properly. Before you analyze or make any changes to your data, you should make sure that the Type setting for each layer is correct.

Tip: You can also edit your layers from the Navigator's Layers list. (Additional tasks, such as setting the active layer and controlling polarity, can also be performed there.)

To add a layer

1. Click on the Name cell for an unused layer number.
2. Type in a descriptive name for the layer. This name can be changed at any time.
3. Click on the Type cell for the layer, and a shortcut menu appears.
4. Select the type of data that the layer contains. This is referred to as "tagging" the layer. It is important to properly tag each layer, as several VisualCAM functions utilize the layer type for proper operation. If the layer is an NC layer, you are prompted to select the NC Tool Table to associate with the layer.

   When a layer is empty, it is considered "unused". The layer name and type will appear in a light-gray color until data has been added to the layer.

Tip: The number of available layer numbers in the layer table depends upon the Max Layers setting in the Options|Configure command. If you want to add a layer but there is no unused layer available, increase the Max Layers number.

To make a layer visible/invisible

All layers that are currently visible in the workspace have a check mark next to their Layer number. Visible layers can be edited, invisible layers cannot be edited. You can also control layer visibility with the Color Bar.

- To make a layer visible, click on an empty box to place a check in it. To make a layer invisible, click on the box to remove the check from it.

To delete a layer

1. Click on the layer number of the desired layer.
2. Click the Cut button.

   When you click OK to close the Layer Setup dialog box, you will be warned that you have cut layers that were not pasted elsewhere in the layer list. Click Yes to confirm your deletion, and the layer will then be removed.

To reorder layers

1. Click on the layer number of a layer you wish to move.
2. Click the Cut button.
3. Click on the number of a layer that you wish to move the cut layer before or after.

4. Click the Paste Above button to move the cut layer before the selected layer, or click on the Paste Below button to move the cut layer after the selected layer. All affected layers are automatically renumbered, as appropriate.

To change the draw or flash colors used on a layer

The Flash and Draw cells for each layer control how the draws and flashes appear in the workspace. You can also control layer colors with the Color Bar.

1. To change a color for a particular layer, click on the desired Flash or Draw cell. A Color Selector appears.
2. Click on the color you wish to use. To dismiss the Color Selector without changing the color, press the Esc key.

To change the color choice at a particular location in the Color Selector, right-click on the desired color. This opens the standard Windows color selector that allows you to define a color.

If you have a specific set of layer colors you would like to use for all your designs, you are provided with a file in the Macros folder called DEMO.MAC, which includes a macro called SetLayerColors. Use the following steps to utilize this macro.

1. Open the DEMO.MAC file the Macro Developer, and edit it to specify your desired colors. Save the file and return to the main VisualCAM window.
2. Select the Macro|Load command and load the DEMO.MAC file.
3. Select the Macro|Run command, and run the SetLayerColors macro.

Tip: If you wish to run the SetLayerColors macro automatically when VisualCAM starts, rename it to OnStartup. To run it whenever you select the File|Open command, rename it to OnOpen.

To add comments to a layer

You can add comments to a layer for your reference. If you export your data as Gerber, these comments will be added to the files (preceded by a "G04").

1. Click on the number of the desired layer.
2. Click the Comments button. A text editor appears.
3. Type your comments into the text field. You can cut, paste, and delete text using the Edit menu.
4. Select the File|Save command in the text editor to save your comments.

Setup|Apertures

Toolbar Button: 

Equivalent Hotkey: Shift+A

This command allows you to edit the aperture list, which appears on the left side of the dialog box. You can view your aperture list in two ways, and modify it according to your needs.

VisualCAM supports 26 intrinsic aperture types, with full support for ODB++ intrinsic apertures.

Following are detailed descriptions of each feature in the Aperture Setup dialog box. For step-by-step instructions on creating and modifying apertures, see the *Working with Apertures* tutorial.

Aperture List Views

D-Code View

This tab presents a complete table of the apertures sorted by D-code number. The Properties column provides you with the size and simple graphical representation of the shape of the aperture. The DCode column provides you with the D-code number. Apertures lists start with D10, as the number D1-D9 are reserved for photoplotter command codes, and are not considered valid aperture numbers by VisualCAM. The Angle column indicates if the aperture has been rotated (a positive number indicates counter-clockwise rotation. A negative number indicates clockwise rotation). The Type allows you to specify if the aperture represents SMT, a through-hole pad, or a thermal. Click on the desired column heading to sort the list by that attribute. Click again to reverse the sort order. Note that the sorting is purely for viewing -- item D-codes are not changed.

If a D-code is not used in the current design, its D-code number and Angle are gray. Used D-codes have black text.

Shape View

Allows you to view the apertures sorted by their shape. A plus (+) icon indicates that apertures of a particular shape exist. You can expand the "tree" for a particular shape by clicking on the + icon. The apertures for each shape are sorted by size, and you can view the D-code number, degrees of rotation (if applicable), and type by further expanding the tree. All the information in this view can be edited by right-clicking on the desired property.

Quick Add
Allows you to simply type in D-code information in an ASCII list. This is useful if you, for example, have a hardcopy aperture list that you want to quickly enter into VisualCAM. Simply type in the information as indicated: D-code Shape size1 [size2 [angle]]. If no angle is given, then no rotation is assumed (you do not need to type in 0 for the angle value if there is no rotation). The Shape value does not have to be completely spelled out; the first 1 to 3 letters will suffice. To avoid any confusion the following usage is recommended:

Bullet `Bul [width] [height] [angle]`

Butterfly `But [width] [angle]`

Butterflies are created in a square shape, by default.

Diamond `Di [width] [height] [angle]`

Donut `Do [outer] [inner]`

Donuts are created in a round shape, by default.

Ellipse `E [width] [height] [angle]`

Hexagon `H [width] 0 [angle]`

The height is automatically calculated for you, as equilateral hexagons are created by default. The 0 is used as a placeholder if you need to specify an angle.

Moiré `M [outer ring diameter] 0 [angle]`

A simple moiré with two rings is created. The cross-hair lines are calculated using the outer ring diameter multiplied by 1.4. The 0 is used as a placeholder if you need to specify an angle.

Oblong `Ob [width] [height] [angle]`

Octagon `Oc [width] 0 [angle]`

The height is automatically calculated for you, as equilateral hexagons are created by default. The 0 is used as a placeholder if you need to specify an angle.

Rectangle `Re [width] [height] [angle]`

A rectangle with square corners is created, by default.

Round `Ro [diameter]`

Square `S [width] 0 [angle]`

The 0 is used as a placeholder if you need to specify an angle.

Thermal `Th [outer diameter] [inner diameter] [angle]`

A round, 4 spoke thermal is created by default. The air gap is calculated as: ([outer diameter] - [inner diameter]) / 2.0

Triangle `Tr [width] [height] [angle]`

Shape Tabs
These tabs represent aperture shapes. Each has its own set of parameters.

Bullet
Enter a horizontal Width and vertical Height for the bullet. Note that you can rotate this aperture, by entering an Angle value.

Butterfly
Select whether you wish the butterfly to have Round or Square ends, and define the horizontal Width of the butterfly. Note that you can rotate this aperture, by entering an Angle value.

Custom
To assign a custom aperture to a D-code, select the custom aperture from the Custom Aperture Name list. If you wish to create or modify a custom aperture, use the Custom Ap button. Note that you can rotate this aperture, by entering an Angle value.

Diamond
Enter a horizontal Width and vertical Height for the diamond. Note that you can rotate this aperture, by entering an Angle value.

Donut
Select the Shape of your donut, and define its Outer diameter and Inner (hole) diameter. Note that you can rotate this aperture, by entering an Angle value.

Ellipse
Define the horizontal Width and the vertical Height of the ellipse. Note that you can rotate this aperture, by entering an Angle value.

Hexagon
Define the horizontal Width and the vertical Height of the hexagon. Enter a Corner Size (as illustrated below), or click the Equilateral button to automatically create an equilateral hexagon. Note that you can rotate this aperture, by entering an Angle value.



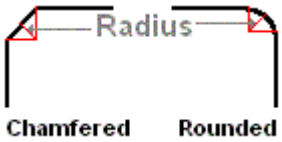
Moiré
Specify the number of rings in the Count box, the Width of each ring, and the Gap or space in between. Then define the Width and Length of the cross-hair lines. Note that you can rotate this aperture, by entering an Angle value.

Octagon
Define the horizontal Width and the vertical Height of the octagon. Enter a Corner Size (as illustrated below), or click the Equilateral button to automatically create an equilateral octagon. Note that you can rotate this aperture, by entering an Angle value.



Oblong
Define the horizontal Width and the vertical Height of the oblong. Note that you can rotate this aperture, by entering an Angle value.

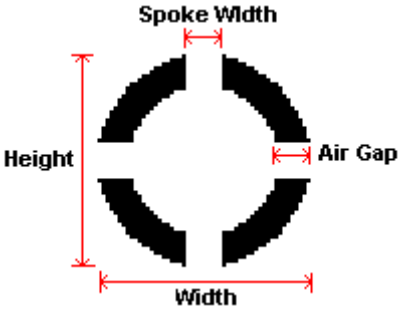
Rectangle
Define the horizontal Width and the vertical Height of the rectangle. The corners can be Square (right angles), Rounded, or Chamfered. For rounded or chamfered rectangles, you specify which corners are rounded/chamfered, and the radius of the edges (as illustrated below). Note that you can rotate this aperture, by entering an Angle value.



Round
Define the Diameter of the round aperture.

Square
Define the Width, or length of the sides, of the square aperture.

Thermal
You can create either a Round, Square, or Rectangular thermal. Specify the horizontal Width of the thermal, and the Height if you are creating a Rectangle. The Air Gap Width is the thickness of the thermal's "line" and the End Style defines the appearance of each of the line ends at the spokes. Specify the number of spokes in the Count box, any rotation Angle for the spokes, and their Width. By definition, round thermals have round centers, and rectangular thermals have rectangular centers. Square thermals can have either.



Triangle
Select the Width of the base of the triangle, and its Height. Note that you can rotate this aperture, by entering an Angle value.

Preview Display

The display in the upper-right corner shows the currently selected aperture. It is a dynamic representation of the aperture's appearance, but not its actual size.

Buttons

Arrows
Located underneath the shape tabs, these arrows allow you to move sequentially through the aperture list views, by D-code number.

Current D-code
The currently selected D-code number is displayed next to the arrow buttons.

Apply
Applies the currently defined values to the D-code selected in the aperture list. You can have the system automatically apply changes to selected D-codes by clicking the down-arrow button, and selecting the Auto Apply setting.

Reset
Reverts a D-code's properties to the most recently saved values.

Units
This button opens the Configuration dialog box, which allows you to change the units of measure. Any changes you make to this dialog box will be reflected in Aperture Setup dialog box when you click OK.

Find D-Code
This button allows you to quickly jump to a particular D-code in the list

Find Text
This button allows you to search for a D-code that contains the text string you specify. Any text appearing in the aperture list may be searched for. For example, you could type **.05** to find the next occurrence of a 50 mil aperture.

Add D-Code
Click this button to add a new D-code to the aperture list. There are two ways to add a D-code.

- Select an existing D-code, alter its values as necessary, then click the Add D-Code button to add a new D-code with those values.
- Click the Add D-Code button to add a D-code with no size, define the D-code's size and shape, then click the Apply button.

You will be prompted for the new D-code number. The next available number is automatically provided, which you accept or change. You can also add multiple new D-codes by using the following convention: **n,n** adds D-codes using the numbers you specify (**12,14** adds D-codes 12 and 14), and **n:n** adds a range of D-codes (**12:14** adds D-codes 12, 13, and 14).

Delete D-Code
To delete a D-code, select the desired D-code in the list, and click the Delete D-code button.

Compact
This button removes unused and redundant apertures within an aperture list. Each layer then has its D-codes remapped accordingly.

Custom Ap
Click the Custom Ap button to display a menu containing all the utilities necessary for creating and modifying custom apertures.

Edit
The Edit command opens a dialog box with a list of defined custom apertures. A preview of the selected custom aperture appears on the right side. Select the custom aperture you wish to edit, and click OK. The Custom Aperture Editor appears.

Copy
If you want to create a new custom aperture, based on an existing custom aperture, select the Copy command. A dialog box appears, with a list of defined custom apertures. A preview of the selected custom aperture appears on the right side. Select the custom aperture you wish to copy, and click OK. You are asked to provide a new name for the copy. Type in the name and click OK. The new custom aperture is saved. If you wish to modify it, select the Edit command from the Custom Ap menu.

New
To create a new custom aperture, select the New command. You are asked to assign a unique name to the custom aperture, and click OK. The Custom Aperture Editor appears.

Delete
To delete an existing custom aperture, select the Delete command. A dialog box appears, with a list of defined custom apertures. A preview of the selected custom aperture appears on the right side. Select the custom aperture you wish to delete, and click the OK button. The custom aperture is deleted.

Rename
Select the Rename command to change the name of an existing custom aperture. A dialog box appears, with a list of defined custom apertures. A preview of the selected custom aperture appears on the right side. Double-click on the custom aperture you wish to rename. A cursor now appears at the end of the custom name. Type in the new name, as desired, and click OK.

Scale
To increase or decrease the size of a custom aperture, using a scale factor, select the Scale command. A dialog box appears, with a list of defined custom apertures. Select the custom aperture you wish to scale, and click OK. You are then prompted for a scale factor. The X-Size and Y-Size values are multiplied by this number (you cannot enter zero or negative numbers). For example, if you have a custom with an X and Y size of .06, a scale factor of 2.0 increases the sizes to .12. If you choose a scale factor of .5, the sizes decreased to .03.

Flatten
Custom apertures are often made up of a combination of lines and circles, and positive and negative data. Some photoplotters have a problem handling the complexity of some custom apertures, so converting the custom aperture to polygonal data helps in this situation. To convert a custom aperture to one or more polygons, select the Flatten command. In the Select Custom Apertures To Flatten dialog box, select one or more custom apertures to flatten (to select more than one, press the Shift or Ctrl key, when you click on the name of each custom aperture). Click OK to flatten the selected custom apertures. The D-code association and instances of the custom apertures within your design remain as they were, but their "internal structure" becomes that of polygons.

Load From Lib
To load custom apertures from a library file, select the Load From Lib command. In the Open dialog box, select the .vlb or .vcam library file that contains the custom apertures you wish to load, and click OK. A list of the custom apertures in the library file appears. Select the custom aperture(s) you wish to load (use the Ctrl or Shift key for multiple selections) and click OK. If there are any name conflicts, VisualCAM warns you and assigns a new name to the incoming custom aperture.

Save To Lib
To save custom apertures to a library file, select the Save To Lib command. In the Save dialog box, select an existing .vlb library file (if you wish to replace the file), or type in a new library file name, and click OK. A list of custom apertures appears. Select the custom aperture(s) you wish to save (use the Ctrl or Shift key for multiple selections) and click OK. Your library file is saved. If you saved to an existing library file, the customs are *not* merged into the file. All original data within the chosen library file is cleared before saving.

Related topics

Gerber for Beginners

Setup|Composites

Toolbar Button: 

This command allows you to create and modify composites. Creating a composite does not create any layers; it creates composite layer *sets* from existing layers.

Warning: If you plan to export your data in the 274-D or DXF format, create composites or use clear polarity. 274-D and DXF do not support polarity.

To create a composite

1. Create or load the layers that you wish to be part of the composite.
2. Select the Setup|Composites command. The Composite Setup dialog box appears.
3. The Composite list shows any existing composites, and the layers contained within them. Click the New Composite button. A new composite, with a default name (such as "Composite 1") appears in the Composite List.
4. To add a layer to a composite, click and drag the desired layer from the Layer List to the composite name in the Composite List.
5. The layer's polarity is shown in parentheses next to the layer name. Dark is a positive layer, clear is a negative layer. To change a layer's polarity, right-click on a layer within a composite to select a different polarity.

   If you need to remove a layer from a composite, click and drag the layer from the composite back to the Layer List. Click the Delete Composite button to delete an existing composite (This button does *not* delete any data or remove any data layers. Only the association to other layers is removed.)

6. When you are finished, click OK. In the Color Bar and Layer Table, your layers are now tagged as "Composite".

Setup|Layer Sets

The Layer Sets selection displays a menu of items that allow you to setup different kinds of layer sets.

Setup|Layer Sets|View/Edit

Equivalent Hotkey: Shift+L

This command allows you to create or change layer sets. Layer sets allow you to simultaneously change the visibility state of multiple layers.

To create a layer set

1. Create or load the layers that you want to be part of the layer set.
2. Select the Setup|Layer Sets|View/Edit command. The Layer Set Setup dialog box appears.
3. The Layer Set List shows existing layer sets and the layers contained within them. Click the New Layer Set button.
4. Type a name for your layer set in the New Layer Set Name dialog box, and click OK. The layer set appears in the Layer Set List.
5. To add a layer to the layer set, click and drag them from the Layer List to the layer set name in the Layer Set List.

   If you need to remove a layer from a set, click and drag the layer from the set back to the Layer List. Click the Delete Layer Set button to delete an existing layer set (This button does *not* delete any data or remove any data layers. Only the association to other layers is removed.)
6. Select the Enable Color Bar To Control Layer Sets option to allow toggling of the visibility of layer sets from the Color Bar. When you turn one layer in the layer set on or off, all the others in the layer set are also turned on or off.
7. When you are finished, click the OK button.

Setup|Layer Sets|Blind/Buried

Equivalent Hotkey: Shift+B

This command allows you to create layer sets to define the pairing of NC layers to circuit layers for blind and/or buried vias. If your design includes blind/buried vias, the relationship of NC layers to circuit layers must be defined before generating a netlist using the Netlist Generate command.

Tip: The Enable Color Bar To Control Layer Sets option is only available for normal layer sets, created with the Setup|Layer Sets|View/Edit command.

To create blind/buried layer sets

1. Create or load the layers that contain the blind/buried vias.
2. Select the Setup|Layer Sets|Blind/Buried command. The Layer Set Setup For Blind/Buried Vias dialog box appears.
3. The Layer Set List shows existing blind/buried layer sets and the layers contained within them. Click the New Layer Set button.
4. Type a name for your layer set in the New Layer Set Name dialog box, and click OK. The layer set appears in the Layer Set List.
5. To add a layer to the layer set, click and drag them from the Layer List to the layer set name in the Layer Set List.

   If you need to remove a layer from a set, click and drag the layer from the set back to the Layer List. Click the Delete Layer Set button to delete an existing layer set (This button does *not* delete any data or remove any data layers. Only the association to other layers is removed.)
6. When you are finished, click the OK button.

Setup|Layer Sets|MCM/LTCC Stackup

While the layer table can be an arbitrary ordering of layers, when you extract a netlist for MicroChip Modules or Low Temperature Co-fired Ceramic boards, you need to tell the system what the exact stackup of layers is. This function allows you to specify the sequence of electrical and insulating layers.

To define MCM/LTCC stackup

1. Select the Setup|Layer Sets|MCM/LTCC Stackup command. The system looks at all conductive, NC, or insulator layers and automatically attempts to stack the layers. The automatic ordering will be the first conductive layer found, then the first NC/insulator layer found, then the next conductive layer, and so on, until no more NC/insulator or conductive layers are found.
2. To change the layer order, click on the incorrect layer, then select the correct layer from the drop-down list.

If you need to clear your choices and start the ordering over again, click the Clear button (which then turns into the Reset button), and click the Reset button. The order is set to the system default.

3. Click OK when you are finished.

Setup|NC Tools

Toolbar Button: 

Equivalent Hotkey: Shift+T

This command opens the NC Tool Table, which allows you to edit the drill and mill tools used in your NC layers. You can define tools in multiple tool tables, which can be designated as a "Drill" table (only drill tools are used) "Mill" table (only mill tools are used), or "Both" (drill and mill tools are combined on a single table).

To view/change the units of measure

- Click the Units button. The Configuration dialog box appears, with the Units and Precision tab active. Changing the units of measure here affects how measurements are represented to you.

To add a new tool

- Click the Add Tool button. A new tool is created, using the next available tool number. Define your tool, as needed (see the definitions of the various tab options below).

To delete a tool

1. Click on the number of the tool you wish to delete.
2. Click the Delete Tool button. The tool is immediately deleted.

Warning: You cannot delete a tool that has been used.

To compact the tool table

- To renumber the tools so that they are consecutive, click the Compact button. You are also given the option to delete all unused tools from the tool table, before compacting it.

To save the tool table

1. To save your tool list as a .nct file, click the Save button. The NC Tool Table Files dialog box appears.
2. Specify a file name for your tool list, and click Save. You can load this file in future databases.

To load a tool table

If you have a tool table already loaded, and its tools are used in the design, loading another tool table will make both tables available. If none the existing tool table's tools are used, loading another tool table will replace the existing table.

1. To load a previously saved .nct file, click the Load button. The NC Tool Table Files dialog box appears.
2. Select the file you wish to load, and click Load. The table appears in the NC Tool Setup dialog box.

Tip: To load a tool list that was created by another program, use the File|Import|NC Tool List command.

To merge a tool table file with an existing table

1. Click the Merge button. The NC Tool Table Files dialog box appears.
2. Select the .nct file you wish to merge in, and click the Open button. If you have tools that share the same tool number as the existing table, the NC Tool Mapping dialog box appears.
3. The system attempts to map tools with similar properties to each other (this is also done by clicking the Map button). The existing tool is shown first, then the incoming "mapped" tool is linked to it underneath. To change the assignment manually, click on the mapped tool and drag it to a different existing tool. You can then select the Overwrite Existing Tool Data Of Mapped Tools option, if you wish to overwrite the existing tools with the mapped tools.

   -or-

   You can have the system automatically assign the tools, by using one of the other three Automatic Tool Mapping Algorithms. Click the Overwrite button if you wish to overwrite the existing tools with the mapped tools. Click Ignore to ignore any incoming tools that are mapped to existing tools. Click Add to map the incoming tools to new, unused tool numbers.
4. When you are satisfied with the tool assignments, click OK.

Too add a new tool table

- Click the Add Table button. An empty table appears, with a default name. Define your tool table and tools, as needed (see the definitions of the various tab options below).

To rename a tool table

1. Click the Rename Table button. The Rename NC Tool Table dialog box appears.
2. Change the name as desired, and click OK.

To delete a tool table

- Click the Delete Table button. You can only delete a table whose tools are not currently used in the database. If any tools are used, the button is unavailable.

Design Data

The Design Data tab contains the information that normally comes from a CAD system or the CAD system's NC file. Following are descriptions of the data contained in each column.

Number

This is the tool number. The symbol next to the number indicates the type of tool it is.



Size

The tool diameter. The units of measure are defined by the Options|Units/Precision command.

Type

The type of tool, which controls whether a tool is used for drills, mill paths, or both. The type of tool that can be defined depends upon the Type selected for the tool table. A tool defined as "Both" is a tool that can create either drill or mill paths, and can therefore be used in any tool table.

Usage

Whether the tool is used to create plated, non-plated, or both types of hits.

Qty

The number of times the tool is used in the currently loaded database.

When viewing a tool table for a panel, the Qty amount will be "0" for every tool. That is because the panel images, along with the tool mapping for the images, are "virtual". The Qty can only be viewed if the panel is flattened.

Legend

One or two alphanumeric characters that are used to designate the tool in a drill hole chart.

Mask Size

The mask size is used by the Tools|NC|Drill Mask command to generate drill mask information for tenting non-plated holes on primary NC layers.

Pilot Tool

If a tool is particularly large, it may cause damage to a board without a pilot hole. You can select an existing, smaller tool to create the pilot hole.

Export Order

Normally, tools are output by tool number in the NC file, which in turn controls the order in which drill hits or mill paths are created. The Export Order definition allows you to export tools (and their associated hits/paths) in a different order in the NC file. This allows the fabricator to accommodate special tool carousel loading requirements.

Color

You can display NC data in the workspace either by layer color or tool color (you can control this with the Tools|NC|Display Settings command). The Color parameter here defines the tool color.

Pattern

You can associate an existing custom aperture with a tool. If you import stepped & repeated NC data, the individual pattern is converted to a custom aperture, which is then assigned to a tool. You can also define the custom in the Setup|Apertures or Tools|Convert|To Custom command, or load it from a custom aperture library file. This allows you to create a step and repeat pattern of NC data. When a custom aperture is associated with a tool, the data within the aperture definition are analyzed and the tools necessary for the data are added to the tool table (if they do not already exist). If there are conflicts with the tools needed in the custom aperture definition and those already existing in the table, a mapping dialog box appears so that you can resolve the conflict.

Export Data

The Export Data tab allows you to apply parameters that are specific to the NC equipment being used in your fabrication shop. (In some situations, a CAD system may support some of these extended parameters, such as feed or speed.) The Export Data information is not always required prior to exporting your database, as it is typically assigned during the tooling process by your fabrication shop's CAM operator. For any value that is set to "Default" or "0", no information is exported, and therefore whatever is programmed into the machine is used. If in doubt, leave the Export Data information set to its default settings, or check with your fabrication shop.

Following are descriptions of the data contained in each column.

Number
This is the tool number. The symbol next to the number indicates the type of tool it is.

Drill
Mill
Both

Size
The tool diameter. The units of measure are defined by the Options|Units/Precision command.

Plunge
The in-feed rate of the tool, in either inches per minute or mm per second (depending upon your units of measure). A value of "0" or "Default" uses the machine's setting.

Retract
The extraction rate of the tool, in either inches per minute or mm per second (depending upon your units of measure). A value of "0" or "Default" uses the machine's setting.

Feed
The rate at which the mill tool travels through the material to create the paths, in either inches per minute or mm per second (depending upon your units of measure). Specify "Default" or "0" to use the machine default values. You can override this setting on a path-by-path basis, using the Edit|NC|Path Properties command.

RPM
The spindle speed in rotations per minute. Specify "Default" or "0" to use the machine default values.

Depth Offset
The "mean depth" is how deeply a tool cuts into the material. This is set at the machine, as it is a function of the thickness of the back-up material and the style of tools used. A positive depth offset causes the drill or mill tool to cut into the material above the mean depth, and a negative depth offset causes the drill or mill tool to cut further into the material than the mean depth.

Max Hits
The maximum number of drill hits a tool may be used for, before it must be replaced. This setting is only available for tools designated as "Drill" or "Both".

Link Tool
If the tool is broken or the Max Hits limit is reached, this is the number of the tool is used as a replacement.

Compensation Index Table

If you wish to apply compensation to any mill paths, the Compensation Index Table controls the amount of compensation used, based on the tool that is used to create the paths. The Index column shows the tool numbers, and the Compensation Value shows the compensation applied to paths created by that tool. By default, compensation is the tool's radius. If the compensation is 0, then the machine-set compensation value is used.

You can control whether compensation is applied to a mill path by selecting the Edit|NC|Path Properties command.

Setup|Break Tabs

Toolbar Button: 

The Setup|Break Tabs command is used to define any break tabs that you wish to use in your mill paths. If you frequently use the same tabs, you can save the tabs to a file for use on future jobs.

If you plan to add a break tab to a compensated mill path, you should define the tab with the compensation in mind (i.e. how you want the tab to appear after compensation has been applied).

If you add a break tab to a mill path, then change the definition of that tab in the Setup|Break Tabs function, any occurrences of the tab in your database will change to reflect the new definition. If the new definition results in the tab being larger, you will be warned if your changes create an invalid tab at any location. If you choose to keep the changes, any invalid tabs are deleted. Otherwise, you can return to the setup dialog box and modify the tab again.

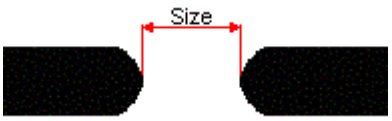
To change the appearance of an individual break tab, define a new tab here (if an appropriate tab does not already exist), then use the Edit|NC|Change|Break Tab command.

To define a new break tab

1. Select the Setup|Break Tabs command. The Mill Path Tab Setup dialog box appears.
2. Click the Add Tab button.
3. In the dialog box that appears, specify a new tab name (or keep the default name), and click OK. The tab now appears in the list box, and is given some default properties. A preview of the tab appears to the right of the list. This preview changes as you define your tab, so that you can achieve accurate results.

4. Click the plus (+) button next to the tab name, to display its properties.
5. There are five types of break tabs, and the properties displayed depends upon the type. By default, the tab you added is a simple Break. To change the type, click on "Break" to display a list. Following are descriptions of the different types of tabs, and the parameters for each.

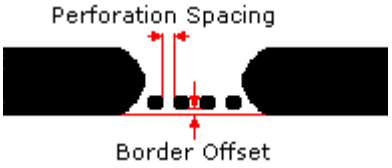
Break
When you create a simple Break, you define the Size, which is the distance between the narrowest points of the two mill path ends. You also specify the Side, which is the side of the path (left or right) that the board is on, relative to the direction the router is traveling.



Stitched Break
A Stitched Break includes small perforation holes between the ends of the mill path. The Side is the side of the path (left or right) that the board is on, relative to the direction the router is traveling. The Side also determines the "border" that the perforation holes are placed relative to. The borders are the edges that would have been created if the router had continued to cut the material. The perforations are placed on the border closest to the board.

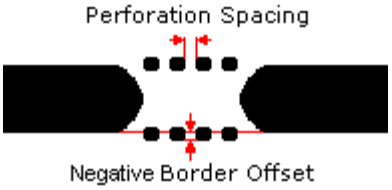
The Calculated Size of the break (the distance between the narrowest points of the two mill path ends), is determined by the other parameters:

- **Number of Perforations** is the number of perforation holes to create.
- **Perforation Size** is diameter of the individual perforation holes.
- **Perforation Spacing** is edge-to-edge distance between the holes themselves, and from the mill path.
- **Border Offset** is the distance from the outermost-edge of the perforation holes to the border. If this number is negative, the perforations will be created outside the mill path (inside the board).



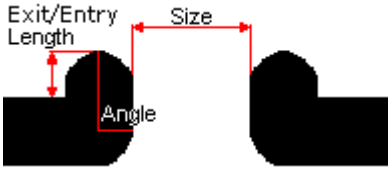
Double Stitched Break
A Double Stitched Break is like a Stitched Break, except there are perforations on *both* borders of the break. The Side is the side of the path (left or right) that the board is on, relative to the direction the router is traveling. The Calculated Size of the break (the distance between the narrowest points of the two mill path ends), is determined by the other parameters:

- **Number of Perforations** is the number of perforation holes to create.
- **Perforation Size** is diameter of the individual perforation holes.
- **Perforation Spacing** is edge-to-edge distance between the holes themselves, and from the mill path.
- **Border Offset** is the distance from the outermost-edge of the perforation holes to each border. If this number is negative, the perforations will be created outside the mill path.



Crown Break
A crown break creates angles in the mill path, at the break. The Side is the side of the path (left or right) that the board is on, relative to the direction the router is traveling. The Side also determines where the exit/entry "legs" are located; the legs are angled away from the board.

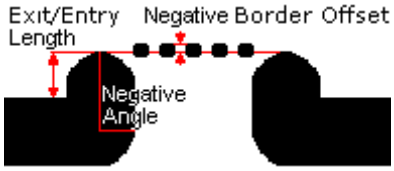
The Size is the distance between the narrowest points of the break. The Exit/Entry Length is the length of the legs. The Exit/Entry Angle is the angle of the legs, relative to the mill path, in degrees. This can be any value between -90 and 180. A negative value means that the legs cut into the board.



Stitched Crown Break
A Stitched Crown Break is a combination of a Stitched Break and a Crown Break; it has both perforations and angled exit/entry points. The Side is the side of the path (left or right) that the board is on, relative to the direction the router is traveling. The Side also determines where the perforations and exit/entry legs are located; the perforations are placed on the border closest to the board, and legs are angled away from the board.

The Calculated Size of the break (the distance between the narrowest points of the exit/entry points), is determined by the other parameters:

- **Number of Perforations** is the number of perforation holes to create.
- **Perforation Size** is diameter of the individual perforation holes.
- **Perforation Spacing** is edge-to-edge distance between the holes themselves, and from the mill path.
- **Border Offset** is the distance from the outermost-edge of the perforation holes to the border. If this number is negative, the perforations will be created outside the mill path (inside the board).
- **Exit/Entry Length** is the length of the legs.
- **Exit/Entry Angle** is the angle of the legs, relative to the mill path, in degrees. This can be any value between -90 and 180. A negative value means that the legs cut into the board.



6. Add additional tabs, as desired, by repeating steps 2-5.

Tip: The Delete Tab button is used to delete a break tab definition. If you have added a break tab to a mill path, you will be asked if you want to delete all occurrences of the tab as well. If you do not want to delete all occurrences of the tab, you must first use the Edit|NC|Change|Break Tab command to change the occurrences of the tab to a different one, then you can delete the tab definition. (If you want to delete a break tab from a mill path, but not delete the tab definition, use the Edit|NC|Delete|Break Tab command.)

7. When you are finished, you can save your tab definitions to a .tab file by clicking the Save Tabs button. (This is not required to save the definitions for the current database -- only if you wish to use them again on future databases.)
8. Click the OK button to exit the dialog box. You may now add break tabs to your mill paths, using the Add|Break Tab command.

Setup|Stencil Shapes

See the Tools|Stencils|Setup Shapes command for a description of this dialog box.

Setup|Footprint Library

See the Assembly|Setup Footprint Library command for a description of this dialog box.

Documentation Menu

The Documentation menu commands allow you to utilize different reporting and redlining functions.



Documentation|Reports

You can generate reports for the apertures and NC tools used in a design.

Documentation|Reports|Apertures

Toolbar Button: 

Select this command to generate an Aperture Report. An aperture report details which D-codes, along with their definitions, are being used on a per layer basis. Included in the report are use counts for both flashes and draws.

Report Field
The body of the dialog box displays the D-codes, shape of the item, width (X-Size) and height (Y-size), whether it is used as a flash or draw, and the number of instances of each. At the bottom a total number of flashes and draws are given. If an aperture has an unknown shape, or is zero in size, it is highlighted for easy recognition.

Edit Selected
Clicking this button opens the Aperture Setup dialog box, with the current aperture set to the aperture highlighted in the report. You may also double click directly on a report item to edit that aperture.

Layer
Allows you to enter the layer for which the aperture information is to be reported. 0, or All, will allow all layers to be considered, while any individual number will generate the aperture information for only that layer. If you change the selected layers, click the Generate button to regenerate the report

Print
Click this button to print the report to the default Windows printer.

Save
Click this button to save the report to a text file, for further manipulation or later printing.

Documentation|Reports|NC Tools

Toolbar Button: 

Select this command to generate an NC Usage Report. This report details which NC tools, along with their definitions, are being used on all NC layers.

Report Field
The body of the dialog box displays a list of each tool that is used, based on your Layer, Table, and Type selections. The tool's Number, Size, Type, Legend, and the number of times it is used are shown, and you can sort the table by any of these items by clicking on the column heading.

Edit Tools
Clicking this button opens the NC Tool Setup dialog box, where you may change the properties of any NC tool. You may also double-click on the Report Field to open the NC Tool Setup.

Layer
To view only the NC tool information for a specific layer, select the desired layer from this option.

Table
Select the tool table that you want to view the usage information for.

Type
Select the type of tool you wish to view usage information for: Drill, Mill, or Both.

Save
Click this button to save the report to a text file.

Print
Click this button to print the report to the default Windows printer.

Documentation|Reports|Embedded Passives

Select this command to generate an Embedded Passive Report. This report provides details about each embedded passive that has been built in the current database. You can click on any of the column headings to sort the list by that value.

This command functions the same as the Tools|Embedded Passives|Report command.

Reference Designator
The unique identifier for each embedded passive.

Layer
The passive layer that the embedded passive resistor resides on.

X Location and Y Location
The X:Y coordinates of the center of the resistor.

Resistance
The resistance value, in Ohms.

Save
Click this button to save the report to a text file.

Print
Click this button to print the report to the default Windows printer.

Documentation|Report|BOM

Toolbar Button: 

After you have finished reverse engineering your data, this function creates a part report that includes bill of material and centroid information. The various columns in the report can be sorted and moved according to your needs.

Tip: This function is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

Report Field
The main area of the Part Report dialog box contains a list of every part instance that has been identified in the database. To sort the list alpha-numerically by a particular column, click on the column heading. To move a column, click and drag the column heading to a new location. You can also resize the columns, as needed, by clicking and dragging the column edges.

Layer
To list only those parts on a particular layer, select that layer from the Layer list.

Foot Print
To list only those parts with a particular footprint, select the footprint name from the list.

Device
If you filtered the list by a particular footprint, and that footprint has different devices associated with it, the Device list becomes available, and you can subsequently list only parts that are of a particular device.

Save
This button allows you to save the report to an ASCII text file. The size and position of columns are maintained (separated by spaces), and the current units of measure are used.

Print
This button allows you to print an ASCII version of the report to the default Windows printer. The size and position of columns are maintained (separated by spaces), and the current units of measure are used.

Documentation|Reports|Fiducials

Toolbar Button: 

This function creates a report of the names, locations, and types of fiducials on your board. The various columns in the report can be sorted according to your needs.

Tip: This function is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

Report Field
The main area of the Part Report dialog box contains a list of every fiducial that has been identified in the database. To sort the list alpha-numerically by a particular column, click on the column heading. You can also resize the columns, as needed, by clicking and dragging the column edges.

Type
By default, all fiducials are shown. To list only fiducials of a particular type, select either Board or Part from the Type list.

Location
To list only those parts on a particular side of the board, select that side from the Location list.

Save
This button allows you to save the report to an ASCII text file. The size and position of columns are maintained (separated by spaces), and the current units of measure are used.

Print
This button allows you to print an ASCII version of the report to the default Windows printer. The size and position of columns are maintained (separated by spaces), and the current units of measure are used.

Documentation|Redline

The functions in the Redline menu are used to add comments and other information that needs to be stored separately from normal layer information. Every layer may have its own associated redline information attached to it.

Documentation|Redline|Add Text

Toolbar Button: 

The Add Text command allows you to insert text into the currently selected layer's redline information. Redline information is "virtual"; it is associated with a particular layer, so its visibility is determined by whether the associated layer is visible, but it does not exist on a layer.

To add redline text

1. Make the layer that you want the text associated with the active layer.
2. Select the Documentation|Redline|Properties command, and specify the properties of redline information for the desired layer.
3. Select the Documentation|Redline|Add Text command. The Add Redlining Text dialog box appears, with the following options:

   Height
   The character height of the text to be added.

   Width
   The character width of the text to be added.

   Rotation
   The angle, in degrees, of each line of text, where 0 is horizontal and 90 is vertical.

   Slant
   The angle of the individual characters in the added text. This can be used to produce italic text and other special effects.

   Line Spacing
   The spacing between lines of text. The value used is a percentage of the actual character height. 1.0 will produce spacing the same height as the characters, 0.5 one half, etc.

   Char Spacing
   The inter-character spacing of the text to be added. 1.0 will produce spacing the same height as the characters, 0.5 one half, etc.

   Mirror
   Selecting this causes the text produced by this command to be mirrored and printed right-to-left, so it can be placed on mirrored Gerber files.

   Font
   You may select any font from the Font dropdown list. This list contains all the TrueType fonts on your system, and the special "GerbTool-Stroke" font. GerbTool-Stroke is a simple font that does not use polygonal data or negative polarity.

   The following is a list of important points to remember when using TrueType fonts:

   - TrueType fonts require the use of polygonal data and a combination of positive and negative polarity. When exporting your data, note that, by nature, DXF does not support negative polarity, and the 274-D file format does not support negative polarity or polygonal data.
   - VisualCAM modifies the layer setup by adding additional composite layers. Any previously generated report files that specify layer numbers will be subsequently out of sync.
   - As TrueType fonts require the use of composites layers, composite viewing must be enabled to view the text properly. While running this command enables this mode automatically, you may use the View|Composites command to toggle this mode on and off.

Text File
To load a file that contains text you wish to insert, click the **Load** button and select the file. This is useful if you have frequently used text.

Clicking the **Save** button saves the contents of the text window to the file named in the Text File box.

Text Entry Field
This area is used to display the text that is to be added. Text is entered into this field by either typing it in, or loading it from a file using the Load button above.

4. When you are finished defining your text, click OK. Your text is now attached to your cursor.
5. Place the text by clicking on the desired location in the workspace.
6. You can place copies of the text in additional locations, by clicking on them, or press the Esc key to end the function.

Related topics

Documentation|Redline|Add Balloon Text
Documentation|Redline|Add Arrow

Documentation|Redline|Add Balloon Text

Toolbar Button: 

This command allows you to insert a balloon with text into the currently selected layer's redline information. Redline information is "virtual"; it is associated with a particular layer, so its visibility is determined by whether the associated layer is visible, but it does not exist on a layer.

To add redline text

1. Make the layer that you want the text associated with the active layer.
2. Select the Documentation|Redline|Properties command, and specify the properties of redline information for the desired layer.
3. Select the Documentation|Redline|Add Balloon Text command. The Add Redlining Balloon dialog box appears, with the following options:

Height
The character height of the text to be added.

Width
The character width of the text to be added.

Rotation
The angle, in degrees, of each line of text, where 0 is horizontal and 90 is vertical.

Slant
The angle of the individual characters in the added text. This can be used to produce italic text and other special effects.

Line Spacing
The spacing between lines of text. The value used is a percentage of the actual character height. 1.0 will produce spacing the same height as the characters, 0.5 one half, etc.

Char Spacing
The inter-character spacing of the text to be added. 1.0 will produce spacing the same height as the characters, 0.5 one half, etc.

Mirror
Selecting this causes the text produced by this command to be mirrored and printed right-to-left, so it can be placed on mirrored Gerber files.

Font
You may select any font from the Font dropdown list. This list contains all the TrueType fonts on your system, and the special "GerbTool-Stroke" font. GerbTool-Stroke is a simple font that does not use polygonal data or negative polarity.

The following is a list of important points to remember when using TrueType fonts:

- TrueType fonts require the use of polygonal data and a combination of positive and negative polarity. When exporting your data, note that, by nature, DXF does not support negative polarity, and the 274-D file format does not support negative polarity or polygonal data.

- VisualCAM modifies the layer setup by adding additional composite layers. Any previously generated report files that specify layer numbers will be subsequently out of sync.
- As TrueType fonts require the use of composites layers, composite viewing must be enabled to view the text properly. While running this command enables this mode automatically, you may use the View|Composites command to toggle this mode on and off.

Text File
To load a file that contains text you wish to insert, click the **Load** button and select the file. This is useful if you have frequently used text.

Clicking the **Save** button saves the contents of the text window to the file named in the Text File box.

Text Entry Field
This area is used to display the text that is to be added. Text is entered into this field by either typing it in, or loading it from a file using the Load button above.

4. When you are finished defining your text, click OK. Your text balloon is now attached to your cursor.
5. Balloons have arrows that point to an item that relates to the text. Click on the location in the workspace where you want the point of the arrow to be.
6. Move the cursor to the location where you want to place the text, and click to place it.
7. You can place copies of the text balloon in additional locations, by clicking on them, or press the Esc key to end the function.

Related topics

Documentation|Redline|Add Text
Documentation|Redline|Add Arrow

Documentation|Redline|Add Arrow

Toolbar Button: 

This command allows you to insert an arrow of specified location, direction and size into the current layer's redline information. Redline information is "virtual"; it is associated with a particular layer, so its visibility is determined by whether the associated layer is visible, but it does not exist on a layer.

To add a redline arrow

1. Make the layer that you want the arrow associated with the active layer.
2. Select the Documentation|Redline|Properties command, and specify the properties of redline information for the desired layer.
3. Select the Documentation|Redline|Add Arrow command.
4. Click on the desired location of the arrow's tip in the workspace. The arrow's tale is attached to the cursor.
5. Move the cursor to the desired location for the tail, and click to place it.
6. You may place additional arrows, or press the Esc key to end the function.

Related topics

Documentation|Redline|Add Text
Documentation|Redline|Add Line
Documentation|Redline|Add Balloon Text

Documentation|Redline|Add Line

Toolbar Button: 

This command allows you to insert line segments into the current layer's redline information. Redline information is "virtual"; it is associated with a particular layer, so its visibility is determined by whether the associated layer is visible, but it does not exist on a layer.

To add a line

1. Make the layer that you want the line associated with the active layer.
2. Select the Documentation|Redline|Properties command, and specify the properties of redline information for the desired layer.
3. Select the Documentation|Redline|Add Line command.
4. Click on the location in the workspace where you wish one endpoint of the line to be. The line is attached to your cursor.

5. Move your cursor to the location of the second endpoint of the line, and click on it.
6. You can create additional lines, or press the Esc key to end the function.

Related topics

Documentation|Redline|Add Arrow
Documentation|Redline|Sketch

Documentation|Redline|Sketch

Toolbar Button: 

The Sketch command allows you to insert freeform drawing into the current layer's redline information. Redline information is "virtual"; it is associated with a particular layer, so its visibility is determined by whether the associated layer is visible, but it does not exist on a layer.

To draw a redline sketch

1. Make the layer that you want the drawing associated with the active layer.
2. Select the Documentation|Redline|Properties command, and specify the properties of the line you want to use to create the sketch.
3. Select the Documentation|Redline|Sketch command.
4. Click and drag the cursor in the workspace to draw any desired shape.
5. When you are finished, press the Esc key.

Related topics

Documentation|Redline|Add Line
Documentation|Redline|Add Arrow

Documentation|Redline|Delete

Toolbar Button: 

The delete command deletes selected redline information.

To delete redline information

1. Select the Documentation|Redline|Delete command.
2. Select an item to delete. Use the Selection Filter as an aid in selecting items, if necessary.
3. When you select an item to delete, you are asked to confirm your selection. If you wish, you can disable the confirmation message by clicking the Disable Prompts button in the Status Bar. This causes each item to be deleted immediately after selection. The button is a toggle, so if you want to enable the confirmation messages again, just click the Enable Prompts button.
4. When you are finished deleting items, press the Esc key.

Documentation|Redline|Properties

Toolbar Button: 

This command allows you to specify the D-code and color used when adding redline information. Redline information is "virtual"; it is associated with a particular layer, so its visibility is determined by whether the associated layer is visible, but it does not exist on a layer.

To define redline properties

1. Select the Documentation|Redline|Properties command. The Redline Properties dialog box appears, with the following options:

   Active Layer
   This allows you to specify the D-code and color used when adding subsequent redline information on the currently active layer. Existing D-code information is not affected by changing these properties.

   In other words, the properties that are actually used when adding redlines are those specified for the Active Layer. This allows you to specify (and save) different colors and D-codes for use in different layers.

   Default
   All redlining is created using the Default settings, unless you override them for individual layers by changing the Active Layer settings. You may change the default line size and/or line color used. The specified line size will be mapped to an appropriate D-code.

2. When you are finished, click the OK button to save your settings.

Documentation|Redline|View Redlining

Toolbar Button: 

The View Redlining command toggles the view of redline information for all of the currently visible layers on and off. When turned on, the redlining information will appear on the screen as well as when printed. Redline information is "virtual"; it is associated with a particular layer, so its visibility is determined by whether the associated layer is visible, but it does not exist on a layer.

Documentation|Drawing

The Drawing menus contain commands that allow you to edit drawing information contained in your design. With these commands you can create quality mechanical drawings and diagrams complete with "intelligent" drawing primitives which automatically update themselves.

Multiple drawings can be contained within the design by first selecting the Setup|Layers command and tagging each layer you wish to have contain a drawing with the Drawing layer type. Then you can create a drawing is by making the desired drawing layer the active layer, then using the appropriate functions found under this menu.

Drawing information, as with all other layer information, is stored in the design file. Drawings can also be plotted and exported in the same manner as other layers.

Documentation|Drawing|Dimensioning

The Dimensioning command presents a menu of dimensioning functions, which display the distance between two items in the design using lines, arrows, and text.

Documentation|Drawing|Dimensioning|Add Dimension

Toolbar Button: 

This command allows you to add one or more dimensions to the current drawing layer. Each dimension reports the distance between two features—i.e. flash center points, and draw endpoints—and/or text that you specify. The color of the dimension lines and text in the workspace depends upon the color you have set for flashes on your drawing layer.

To add a dimension

1. Make the layer that you want the dimensioning associated with the active layer. The layer must be tagged as a Drawing layer (see Setup|Layers).
2. Select the Documentation|Drawing|Dimensioning|Properties command to define the properties of the dimension measurements.
3. Select the Documentation|Drawing|Dimensioning|Add Dimension command. The Add Dimension dialog box appears, with the following options:

   Type
   Linear specifies that the dimension is of two specified items.

   Continued specifies that the first dimension will be of two specified items and that any subsequently specified dimensions will be relative to the last point so that all of the dimensions will appear as connected.

   Baseline specifies that the first selected item will specify the starting point for all subsequent dimensions. Dimensions specified after the first one are automatically offset from the last dimension in the same direction as the dimension text (see Text Spacing).

   Ordinate specifies that dimensions will be created in the same manner as Continued, only with no dimension lines between the extension lines and with the dimension text rotated 90 degrees.

   Orientation
   Horizontal specifies that the new dimension will measure the horizontal distance between two features.

   Vertical specifies that the new dimension will measure the vertical distance between two features.

   Text Spacing
   Specifies the distance to be used when spacing baseline-type dimensions.

   Height
   The character height of the text to be added.

   Width
   The character width of the text to be added.

   Rotation
   The angle, in degrees, of each line of text, where 0 is horizontal and 90 is vertical.

Slant
The angle of the individual characters in the added text. This can be used to produce italic text and other special effects.

Line Spacing
The spacing between lines of text. The value used is a percentage of the actual character height. 1.0 will produce spacing the same height as the characters, 0.5 one half, etc.

Char Spacing
The inter-character spacing of the text to be added. 1.0 will produce spacing the same height as the characters, 0.5 one half, etc.

Location
Left/Over specifies that dimension text is to appear to the left of the dimension (if this is a vertical dimension) or over the dimension (if this is a horizontal dimension).

Middle specifies that dimension text is to appear in the middle of the dimension (the default setting).

Right/Under specifies that dimension text is to appear to the right of the dimension (if this is a vertical dimension) or over the dimension (if this is a horizontal dimension).

The Left/Over, Middle, and Right/Under settings may be overridden if dimension text and/or arrows will not fit between extension lines.

Mirror
Selecting this causes the text produced by this command to be mirrored and printed right-to-left, so it can be placed on mirrored Gerber files.

Font
Only the special "GerbTool-Stroke" font may be used. GerbTool-Stroke is a simple font that does not use polygonal data or negative polarity.

Text File
To load a file that contains text you wish to insert, click the **Load** button and select the file. This is useful if you have frequently used text, such as fabrication information.

Clicking the **Save** button saves the contents of the text window to the file named in the Text File box.

Text Entry Field
When a dimension is created with the text box empty, the dimension text is automatically generated by taking the measured distance and appending the appropriate text specified in the Dimension Properties dialog box. If, however, you specify text in the text box it will be used to specify what to display as the dimension text. Two special sequences may be present in the text. The sequence **<>** (a less-than and greater-than symbol) specifies that the pattern is to be replaced with the distance, using current units. The sequence [] (an open bracket and closed bracket) indicates that the pattern is to be replaced with the distance in metric units if Imperial units are in use, or in Imperial units if metric units are currently in effect.

As an example, consider the following: If Imperial units are currently specified and you put the following into the text box:

```
This distance is <>
```

Assuming the dimension is for features 10" apart and the Dimension Properties dialog box contains the default settings, you would see the following resulting string:

```
This distance is 10"
```

4. Click the Options button to specify other dimension line properties in the Dimension Options dialog box:

Arrow1 Type, Arrow2 Type
Specifies the type of arrows to use in the dimension.

Line Widths
Extension specifies the distance to offset the dimension extension lines from the dimensioned features.

Dimension specifies the line thickness to use when drawing the dimension.

Text specifies the line thickness to use when drawing dimension text.

Arrows Inside Extension Lines
When selected, arrows are drawn within the extension lines (the default case); otherwise, arrows appear outside of the extension lines. Note that this setting may be overridden if the dimension text and/or arrows cannot fit within the extension lines.

Box Around Text
When selected, a box will drawn around the dimension text; otherwise no box will be drawn.

5. Click the OK buttons in the Dimension Options and Add Dimension dialog boxes.

6. Click on the two features to dimension, and the point to place the dimension text.
7. If you wish to add a dimension to additional items, repeat the previous step. Otherwise, press the Esc key to end the function.

If dimensioned items are moved, such as by editing or scaling operations you perform, the dimensions will automatically be updated. Dimensions automatically relocate dimension text and/or the direction of arrows as needed in the event that there is insufficient room to fit these between the dimension extension lines. This feature overrides text location and arrow direction settings that may have been specified in the Add Dimension dialog box.

Related topics

Documentation|Drawing|Dimensioning|Modify Dimension
Documentation|Drawing|Dimensioning|Delete Dimension

Documentation|Drawing|Dimensioning|Delete Dimension

Toolbar Button: 

The Delete Dimension command allows you to delete a dimension.

To delete a dimension

1. Select the Documentation|Drawing|Dimensioning|Delete Dimension command. You are prompted to select a dimension to delete.
2. Click on the dimension in the workspace. It is immediately deleted.
3. Select another dimension to delete, or click the Esc key to exit the function.

Documentation|Drawing|Dimensioning|Modify Dimension

Toolbar Button: 

The Modify Dimension command allows you to change any property of a selected dimension.

To modify a dimension

1. Select the Documentation|Drawing|Dimensioning|Modify Dimension command. You are prompted to select the dimension to modify.
2. Click on the dimension in the workspace. The Modify Dimension dialog box appears. See Add Dimension for a description of dimension properties.
3. When you are finished modifying the dimension properties, click OK. The dimension is immediately updated in the workspace.
4. Click on another dimension to modify, or press the Esc key to exit the function.

Related topics

Documentation|Drawing|Dimensioning|Delete Dimension

Documentation|Drawing|Dimensioning|Properties

This command allows you to specify what text will appear immediately after a reported distance in the dimension text. The text is normally used to indicate the units for the reported distance. Note that these are global settings that will automatically affect all dimensions in the design.

To define dimension properties

1. Select the Documentation|Drawing|Dimensioning|Properties command. The Dimension Properties dialog box appears, with the following options:

   Number Decimal Digits
   Select the number of digits to appear after the decimal point in all measurements.

   If Units Are Imperial
   If Imperial units are used, text appearing here will be appended to all dimension text. The default value is **"** (for "inches").

   If Units Are Metric
   If metric units are used text appearing here will be appended to all dimension text. The default value is **mm**.
2. When you are finished, click OK. If there are dimensions in the workspace, the text is immediately updated.

Related topics

Documentation|Drawing|Dimensioning|Add Dimension

Documentation|Drawing|Dimensioning|Add Line

Toolbar Button: 

The Add Line command allows you to add a construction line to the currently selected drawing layer. The color of the line in the workspace depends upon the color you have set for flashes on the layer.

To add a construction line

1. Make the layer that you want the dimensioning associated with the active layer. The layer must be tagged as a Drawing layer (see Setup|Layers).
2. Select the Documentation|Drawing|Dimensioning|Add Line command. The Add Construction Line dialog box appears.
3. Specify the Type of construction line to draw. A preview of the line appears in the Appearance box.
4. Specify the width of the construction line.
5. Click OK. You are prompted to draw the line.
6. Click on the starting point of the line in the workspace. The line is now attached to your cursor.
7. Move the cursor to the end point of the line, and click to place it.
8. You may add additional lines, or press the Esc key to end the function.

Related topics

Documentation|Drawing|Dimensioning|Modify Line
Documentation|Drawing|Dimensioning|Delete Line

Documentation|Drawing|Dimensioning|Delete Line

Toolbar Button: 

The Delete Line command allows you to delete a construction line.

To delete a dimension

1. Select the Documentation|Drawing|Dimensioning|Delete Line command. You are prompted to select a construction line to delete.
2. Click on the line in the workspace. It is immediately deleted.
3. Select another line to delete, or click the Esc key to exit the function.

Documentation|Drawing|Dimensioning|Modify Line

Toolbar Button: 

The Modify Line command allows you to change any property of a selected construction line.

To edit a construction line

1. Select the Documentation|Drawing|Dimensioning|Modify Line command. You are prompted to select the construction line to edit.
2. Click on the line in the workspace. The Modify Construction Line dialog box appears. See Add Line for a description of construction line properties.
3. When you are finished, click OK. The construction line is immediately updated.
4. Click on another line to modify, or press the Esc key to end the function.

Related topics

Documentation|Drawing|Dimensioning|Delete Line

Documentation|Drawing|Drill

The Drill command presents a menu of functions for creating drill hole charts.

Documentation|Drawing|Drill|Add Hole Chart

Toolbar Button: 

The Add Hole Chart command automatically creates a fabrication drill chart for the selected NC layer. A drill chart contains a title block with the legends used to represent different kinds of drill hits, the size of the drill, whether they are

plated or unplated, and how many hits there are of each. Drill locations are automatically marked using the appropriate legend symbol, which is defined in the NC Tool Table.

Tip: It is not necessary to make manual modifications to drill charts when drill information or units of measure change. Just select the Documentation|Drawing|Drill|Update Hole Charts command to automatically update a drill chart.

To add a hole chart

1. Hole charts are created on drawing layers. Make a Drawing layer the active layer.
2. Select the Documentation|Drawing|Drill|Add Hole Chart command. The Add Hole Chart dialog box appears.
3. Select the NC layer that contains the desired drill information from the Drill Layer box.
4. Define the parameters for your Title Text.

   Drill Layer
   Enter the desired layer you wish to create a hole chart from.

   Height
   The character height of the text to be added.

   Width
   The character width of the text to be added.

   Rotation
   The angle, in degrees, of each line of text, where 0 is horizontal and 90 is vertical.

   Slant
   The angle of the individual characters in the added text. This can be used to produce italic text and other special effects.

   Line Spacing
   The spacing between lines of text. The value used is a percentage of the actual character height. 1.0 will produce spacing the same height as the characters, 0.5 one half, etc.

   Char Spacing
   Allows you to specify the inter-character spacing of the text to be added. 1.0 will produce spacing the same height as the characters, 0.5 one half, etc.

   Font
   Only the special "GerbTool-Stroke" font may be used. This is a simple font that does not use polygonal data or negative polarity.
5. If you want to use title text from an existing file, click the Load button and select the desired text file. The text appears in the text entry field, below.

   -or-

   Enter a title for the hole chart in the text entry field. If you want to save this text for use on another job, click the Save button and specify a file name.
6. Click OK. An outline of the title block is attached to your cursor.
7. Click on the location where you want to place the title block. The drill legends are then automatically added to the drawing layer.
8. Press the Esc key to exit the function.

Documentation|Drawing|Drill|Delete Hole Chart

Toolbar Button: 

This command allows you to delete a drill chart from a drawing layer.

To delete a hole chart

1. Select the Documentation|Drawing|Drill|Delete Hole chart command. You are prompted to select the chart to delete.
2. Click on the title block of the desired drill chart. The entire chart is deleted.
3. Select another drill chart to delete, or press the Esc key to exit the function.

Related topics

Documentation|Drawing|Drill|Modify Hole Chart

Documentation|Drawing|Drill|Modify Hole Chart

Toolbar Button: 

This command allows you to change the title block for drill chart.

To modify a hole chart

1. Select the Documentation|Drawing|Drill|Modify Hole Chart command. You are prompted to select the chart to modify.
2. Click on the title block you wish to change. The Modify Hole Chart dialog box appears. See the Documentation|Drawing|Drill|Add Hole Chart command for a description of the dialog box options.
3. When you are finished, click the OK button. The title block is automatically updated.
4. Select another title block, or press the Esc key to exit the function.

Related topics

Documentation|Drawing|Drill|Delete Hole Chart

Documentation|Drawing|Drill|Update Hole Charts

Toolbar Button: 

Use this command if you have made modifications to your data that affect the drill information. Selecting this command automatically updates all drill charts.

Documentation|Drawing|Fabrication

The Fabrication command presents a menu of fabrication drawing functions.

Documentation|Drawing|Fabrication|Add Note Balloon

Toolbar Button: 

The Add Note Balloon command allows you to add a note balloon to the current drawing layer.

To add a fabrication note balloon

1. Make the layer that you want the note balloon associated with the active layer. The layer must be tagged as a Drawing layer (see Setup|Layers).
2. Select the Documentation|Drawing|Fabrication|Add Note Balloon command. The Add Note Balloon dialog box appears, with the following options:

   Symbol Shape
   Specifies that the note balloon is to have a triangle, circle, or square appear around the text.

   Height
   The character height of the text to be added.

   Width
   The character width of the text to be added.

   Rotation
   The angle, in degrees, of each line of text, where 0 is horizontal and 90 is vertical.

   Slant
   The angle of the individual characters in the added text. This can be used to produce italic text and other special effects.

   Line Spacing
   The spacing between lines of text. The value used is a percentage of the actual character height. 1.0 will produce spacing the same height as the characters, 0.5 one half, etc.

   Char Spacing
   The inter-character spacing of the text to be added. 1.0 will produce spacing the same height as the characters, 0.5 one half, etc.

   Font
   Only the special "GerbTool-Stroke" font may be used. GerbTool-Stroke is a simple font that does not use polygonal data or negative polarity.

   Text File
   To load a file that contains text you wish to insert, click the **Load** button and select the file. This is useful if you have frequently used text.

   Clicking the **Save** button saves the contents of the text window to the file named in the Text File box.

Text Entry Field
This area is used to display the text that is to be added. Text is entered into this field by either typing it in, or loading it from a file using the Load button above.

3. When you have finished defining your note balloon, click OK. You are prompted for the placement of the note balloon.
4. Balloons have leader lines with arrows that point to an item that relates to the text. Click on the location in the workspace where you want the point of the arrow to be.
5. Move the cursor to the location where you want to place the text, and click to place it. If the leader line endpoint appears within the note balloon then no leader line will be drawn.
6. You can place copies of the note balloon in additional locations, by clicking on them, or press the Esc key to end the function.

Documentation|Drawing|Fabrication|Delete Note Balloon

Toolbar Button: 

The Delete Note Balloon command allows you to delete a note balloon.

To delete a note balloon

1. Select the Documentation|Drawing|Fabrication|Delete Note Balloon command. You are prompted to select a note balloon to delete.
2. Click on the desired note balloon in the workspace. It is deleted immediately.
3. Click on any additional note balloons you wish to delete, or click the Esc key to end the function.

Documentation|Drawing|Fabrication|Modify Note Balloon

Toolbar Button: 

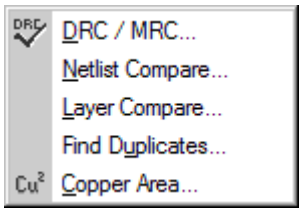
The Modify Note Balloon command allows you to change any property of a selected note balloon.

To edit a note balloon

1. Select the Documentation|Drawing|Fabrication|Modify Note Balloon command. You are prompted to select a note balloon to edit.
2. Click on the desired note balloon. The Modify Note Balloon dialog box appears. See Add Note Balloon for a description of note balloon properties.
3. When you have finished making your modifications, click OK. The balloon is immediately updated.
4. Click on another note balloon to modify, or press the Esc key to end the function.

Analysis Menu

The Analysis menu allows you to perform Design Rules Checking, Manufacturing Rules Checking, and Design For Fabrication analysis, to insure that your design can be manufactured without error.



Analysis|DRC/MRC

Toolbar Button: 

The DRC/MRC analysis function verifies that your design meets the design, manufacturing, and fabrication rules that you specify. Each "pass" defines a set of layers and the design rules that apply to them. You may setup as many passes as required, and execute them all sequentially when you click the Run button. This feature allows precise control of the rules that are applied to your design.

You may also save/load rule sets to external files to create templates that can be used on similar designs. As an aid to those who are analyzing embedded passives, files with embedded passive design rules used by major companies are provided in the RuleSets folder.

Warning: In order to perform this analysis, all layers must be "tagged" with a type. See Setup|Layers if your layers are not tagged. If you want to perform any of the DRC or NLC checks, you must have a valid netlist. Use the Tools|Netlist|Generate command if you do not have an existing netlist. To perform the NLC check, you must also import the netlist you wish to compare against the netlist you generated in VisualCAM.

After performing the analysis, VisualCAM automatically displays the errors in the Navigator, which you may use to view and automatically fix the errors.

If you want to run the analyses again after you have fixed the errors (to verify no errors still exist), you should first regenerate the netlist. This is because any fixes will not be reflected in the netlist that you created before the initial analyses.

Run Name
Each run can be assigned a descriptive name, for easy reference in the future. Each run that is performed is logged in the Navigator, and uses the assigned Run Name. If you do not assign a Run Name, the run is identified by the time and date it is performed.

Highlight Layer
If you would like to have all database items that are involved in analysis errors copied to a new layer, select an empty layer as your Highlight Layer. Select None if you do not want the data copied to another layer. If you select a layer with data, you will be warned before the analysis is run that the layer contains data, and asked if you would like to delete the existing data. This allows you to easily see all the errors at once and print them.

Passes
Each run can contain one or more passes, each with its own checks. Each pass that is performed is logged in the Navigator, under the appropriate run name. You may Add/Delete passes as needed using the buttons. To change the name of an existing pass, click on it to highlight the name, and click a second time. A cursor appears next to the name, and you can then type in a new one.

Layers to Check
Select the layers you wish to analyze.

NC Layer
If you want to perform any analyses on drill data, select your NC layer. Drill checks are not available unless the NC layer is specified.

Window
Select window mode to run your analysis on just a selected area of data. If this option is not selected, the entire design is analyzed.

Don’t Check Items In Select Group
Select this option if you want VisualCAM to ignore all items that are in the current select group. For example, if you have blocks of text that you know you do not want analyzed, you may use the Edit|Select|Add To command to create a select group with these blocks of text. The text will be ignored, thereby producing a much cleaner report.

Max Errors
This text box allows you to set a limit of how many errors can be found/logged. This prevents the analysis from generating a huge report file if you enter incorrect spacing rules. If you do not want to limit the number of errors, click the arrow button and select No Limit (the No Limit command is a toggle).

Analysis Type
Check each analysis you wish to run in the selected pass, and enter the appropriate properties. If a checkmark appears on a folder icon, it means you have selected an analysis to run in that particular category. If an "X" appears next to an analysis, the analysis cannot be run because the appropriate layers have not been selected or do not exist in your design.

Information
- **Min Air Gap** Reports the shortest distance between any two items in the design.
- **Conductive Layer Count** Reports the total number of conductive layers.
- **Board Size** Reports the extents of the Border layer, if one exists. Otherwise the extents of the conductive layers is reported (drawing and other graphical layers are ignored).
- **Hole Count** Reports the tools used on each NC layer, and the number of holes (hits) created by each.
- **Hole Size Count** Reports the number of tools used on each NC layer (does not report tool numbers or hits).
- **Buried Vias** Reports the upper- and bottom-most (start and end) layers for buried vias. Blind and Buried layer sets must be defined in order to run this analysis.
- **Blind Vias** Reports the upper- and bottom-most (start and end) layers for blind vias. Blind and Buried layer sets must be defined in order to run this analysis.
- **Min Pad/Drill** Reports the minimum annular ring spacing from copper pads to plated through-hole drill hits.
- **Min Drill/Drill** Reports the minimum spacing between drill hits.
- **Min NPTH/Copper** Reports the minimum spacing between non-plated through-hole drill hits and copper on external and internal layers.
- **Min Pad/Pad** Reports the minimum spacing from one pad to another for each conductive layer.
- **Min Pad/Trace** Reports the minimum spacing from a pad to a trace for each conductive layer.
- **Min Trace/Trace** Reports the minimum spacing from one trace to another for each conductive layer.
- **Min Pad/Mask** Reports the minimum annular ring spacing from mask openings to their corresponding copper pad, on the top and bottom layers.
- **Min Thermal/Drill** Reports the minimum annular ring from thermal reliefs to plated through-hole drill hits.
- **Min PTH Pad Size** Reports the size of the smallest pad that has been pierced by a plated through-hole drill hit, for each conductive layer.
- **Min Trace Width** Reports the smallest trace width in the design.
- **Min SMT Pitch** Reports the minimum center-to-center spacing of SMD pads. A pad must be identified as SMT in the Aperture List in order to be checked.
- **Legend on PTH Count & Legend on NPTH Count** Reports the number of occurrences of the silkscreen legend touching pads on the top and bottom layers.
- **Missing Hole Count** Reports the number of instances where a through-hole pad on a conductive layer appears to be missing a drill hit.
- **Mill Path Length** Reports the total linear distance each tool travels, per NC layer.
- **Max PTH Registration** This shows the greatest amount of misalignment of data on drill and conductive layers. It reports the maximum spacing of plated through-hole drills (on an NC layer) from their corresponding pads (on a conductive layer).
- **Exposed Copper Count** Reports the number of occurrences of traces exposed by adjacent mask openings.
- **Copper Sliver Count** Reports the number of occurrences of copper slivers. Any areas of copper that are less than the Size you specify will be considered possible copper slivers.
- **Mask Sliver Count** Reports the number of occurrences of mask slivers. Any resist areas that are less than the Size you specify will be considered possible mask slivers.

DRC (Design Rule Check)
- **Pad/Pad** The minimum spacing allowed between pads.
- **Pad/Trace** The minimum spacing allowed between pads and traces.
- **Trace/Trace** The minimum spacing allowed between traces.

- **Border** The minimum spacing allowed between any item and the border.
- **Min Pad** The minimum pad size allowed.
- **Min Trace** The minimum trace size allowed.
- **Stubs** A trace stub is any trace that touches a pad or trace on one end, but does not on the opposite end.
- **Drill/Drill** The minimum spacing required between adjacent drill locations.
- **Drill Annulus** The minimum annular ring required between conductive layers and drill layer.
- **Drill/Copper** The minimum space required between copper entities and drills. Separate tolerances are available for plated through-holes (PTH) and non-plated through holes (NPTH).
- **Pad/Top Mask & Pad/Bot Mask** The minimum annular ring required between top/bottom side pads and the top/bottom solder mask. (Only checks the size of existing openings, not whether there is an opening missing for a pad. Use the Missing Top Mask and Missing Bot Mask checks to find missing openings, if desired.)
- **Drill/Top Mask & Drill/Bot Mask** The minimum annular ring required between the drill layer and the top/bottom solder mask.
- **Missing Top Mask & Missing Bot Mask** Checks for pads on the top/bottom side conductive layer without openings on the top/bottom solder mask.
- **Pad/Top Silk & Pad/Bot Silk** The minimum clearance required between top/bottom side pads and the top/bottom silkscreen layer. (You can use the Tools|Fix SilkScreen command to remove silkscreen data from pads). If you select a "Mask: Yes" property, then the silkscreen layer is instead compared to the corresponding solder mask layer.
- **Missing Drill** Checks for pads that do not have a corresponding drill hit.
- **Coincident Drill** Checks for drill hits that are in the same location but are different sizes.
- **Redundant Drill** Checks for drill hits of the same size that are in the same location.
- **Mill Path** Checks compensated mill pats for errors introduced by the compensation: arcs that implode, paths that cross over themselves, and any break tabs that become invalid because the compensated path is now too short for the tab to fit.

DFF (Design For Fabrication)

- **Acid Traps** An acid trap is an area where etching solution accumulates but does not flow out during manufacturing. This causes over-etching, which hurts yield. This analysis detects potential acid traps in your design. Any areas where acid is likely to accumulate that are smaller than the Size you specify, and any items (traces, pads, etc.) that form an Angle smaller than the one you specify, will be considered possible acid traps.

⚠ *Tip: Because the acid trap analysis looks for areas that are not fully enclosed, it is recommended that you run the resist sliver analysis in conjunction with this, for a thorough check of your design.*

- **Copper Slivers** Copper Slivers are areas of copper that are so narrow that they will likely flake off. This command detects those potential slivers on the selected conductive layers in your design. Any areas of copper that are less than the Size you specify will be considered possible copper slivers.
- **Resist Slivers** Similar to acid traps, resist slivers are small areas of resistive material that have a surface area too small to adhere to the board, and can therefore flake or peel off. Where acid traps are open at one end, resist slivers can be fully surrounded by copper. Depending on your process, the resist slivers test will often find items that are also considered acid traps or pin holes. The Resist Slivers analysis may be more applicable to those using a plate-up photo resist process instead of an etch-back process. Resist areas less than the value you specify are considered resist slivers.
- **Top Mask Slivers & Bot Mask Slivers** Solder mask slivers are areas where the resist is so narrow that they will likely flake off, float, and redeposit themselves in an area that might be soldered later. Any resist areas that are less than the Size you specify will be considered possible mask slivers.
- **Isolated Thermals** Over-etching of surrounding items can result in a thermal being isolated from the rest of a negative plane. This analysis function allows you to oversize the data on the negative plane layers by a specified amount, and determine its effects on the connection of the thermal to the negative plane. You specify the amount of Over Etch that you wish to apply to the data on the specified plane layer.

⚠ *Warning: A thermal can be considered isolated by virtue of its design. For example, if a thermal has ties that are 2 mils wide, and the Over Etch is specified as 2.5, the thermal is isolated.*

- **Starved Thermals** The starved thermal analysis verifies whether each thermal connection to the negative plane is valid, or has been constricted by adjacent data that is too close (or overlapping). You specify the Percentage of the thermal's tie width that can be unblocked by objects. This makes the clearance check relative to each thermal's tie width. The thermal is considered starved if the connection from the interior of the thermal to the exterior, measured

across the minimum distance between the cutouts of the thermal and any other nearby data, is less than the specified value. The Clear value is how many thermal ties must be clear in order for a thermal to be considered not starved. If you specify 0, all ties must be clear. If you select 1, one tie must be clear.

⚠ *Warning: Be aware of the number of ties in all the thermals in your design. If you select 4 as your Clear value, and have thermals with 2 ties, then those thermals would automatically be flagged as starved, even if both of their ties are clear. If you have several different types of thermals, it is recommended that you leave the Clear value at 0.*

- **Top Solder Bridges & Bot Solder Bridges** Openings for pads on a mask layer may be oversized too much, and expose an adjacent trace or another conductive object. The copper for that pad may accidentally get too close during fabrication, and create an unwanted connection, or bridge, to the adjacent object. The Bridge Distance is the distance between the pad and objects in the same mask opening, where the solder can possibly create a bridge. If the adjacent object closer to the pad than this distance, it will be identified as a possible bridge. The Position Tolerance tests for shifting of the placement of mask openings. It oversizes mask openings by the amount you specify, to check if other conductive objects are then exposed by the mask opening. If an object is exposed, and is also within the specified Bridge Distance to the pad, then an error is logged.
- **Pin Holes** A pin hole is a void in an area of solid copper that can cause acid to pool and over-etch the surrounding copper (similar to an acid trap), or can cause resist flakes, potentially causing conductivity problems. Pin holes often result from inaccurate vector-drawn data that leaves tiny voids between the over-lapping draws. You specify the maximum Size of the pin holes to be detected -- areas which are less than this value will be considered pin holes.
- **SMD Pitch** Checks the center-to-center spacing of adjacent SMD pads, to determine if they are too close. A pad must be identified as SMT in the Aperture List in order to be checked.
- **SMD Spacing** Checks the edge-to-edge spacing of adjacent SMD pads, to determine if they are too close. A pad must be identified as SMT in the Aperture List in order to be checked.
- **Layer Registration** Checks your drill layer against the top mask and any other conductive layers you select. If two items are within the tolerance distance you specify, they are considered to be at the same location. The errors reported are those items that, if you use the Edit|Align Layers command or VisualCAM's auto-fix feature to align the layers based on them, will result in 95% or more of the other items on the layer being correctly aligned as well.

NLC (Netlist Compare)

- **Net Short** Multiple external nets touch one internal net.
- **Net Open** Multiple internal nets touch one external net.
- **No Copper** No pad exists at the external net location.
- **No External** No external net point exists at an internal net point.
- **Pad Size Difference** The probe point pad size in the external net differs from the internal net pad size.

⚠ *Tip: The NLC analyses are the same as those performed by the Analysis|Netlist Compare command.*

Embedded Passive

- **Alignment** Checks that the resistor shape is centered (horizontally and vertically) between the termination bars. An error is reported if the resistor shape is out of alignment by more than the specified tolerance.
- **Bar Overlap** Checks that the resistor shape overlaps the two termination bars by the specified amount.
- **Bar Extension** For additive passives, this checks whether the termination bars extend beyond the resistor shape by the specified amount. For subtractive passives, this checks whether the resistor shape extends beyond the termination bar by the specified amount.
- **Bar Width** Checks if the termination bars are at least as wide as the specified amount.
- **Minimum Size** Checks that the active resistor area (not including the overlap) is at least the specified X:Y size.
- **Maximum Size** Checks that the active resistor area (not including the overlap) is no larger than the specified X:Y size.
- **To Copper** Checks the distance of the resistor shape against any adjacent copper. If they are closer than the specified distance, an error is reported.
- **To Drill** Checks the distance of the resistor shape against any drills on the same net. If they are closer than the specified distance, an error is reported.

Properties

These are the defined parameters for each check. To change a property, click on it and type in the value. The property for each analysis type is explained above.

Layer(s)
This column displays the layers used in each check (selected in the Layers To Check column). Some checks, like the Border DRC check, are predefined with the appropriate layer type to analyze (in this example, the layer being "border"), and you then select the layer to compare it against. If no layers have been selected, in cases where one or more conductive layers can be analyzed, **<none selected>** appears. If a layer of the correct type does not exist in your design, **<not defined>** appears. You will need to use the Setup|Layers command to tag the appropriate layer. If a layer of the appropriate type exists but is not selected, **<not selected>** appears.

Save
Click to save current analysis pass/rule settings to an external file.

Load
Click to load pass/rule settings from an external file. This command will load settings from a VisualCAM design file as well as a standalone rule file.

Run
Click to perform selected analyses.

Set As Default
Click to save current settings as the default settings.

Related topics

[Performing DRC & DFF Analyses]

Analysis|Netlist Compare

Graphical netlist comparison allows you to visually compare your design data against the original netlist. VisualCAM performs the comparison automatically, and all violations are stored internally like DRC/MRC and DFF violations. This means you can quickly scan through and fix connectivity issues. This command performs the same comparison as the NLC check, which can be run with other DRC/MRC analyses.

Tip: Several steps should be taken to prepare your data prior to performing the netlist comparison, such as tagging your layers, generating a netlist, and importing the external netlist. See the Netlist Comparison tutorial for details.

Analysis Checks

Shorts Multiple external nets touch one internal net.

Opens Multiple internal nets touch one external net.

No Copper No pad exists at the external net location.

No External Net Point No external net point exists at an internal net point.

Pad Size Differences The probe point pad size in the external net differs from the internal net pad size.

Options to Apply....

Assign External Netnames To Database This option applies the IPC-D-356 net name in place of the VisualCAM Net ID number. The net names will only be applied to those nets that do not contain errors.

Assign Net/Pin Data To UserData Fields This option applies the net names to the User Data for each net, so that you can maintain the net names. This allows you to use VisualCAM commands, including the Query|Item and Query|Net commands, to examine and manipulate the true reference designators, pin numbers, etc.

Add Test Point Pad Layers This creates layers with pads representing probe point data. The information is graphical only, and you must specify the empty layers that you wish to place the pads on. These pads are based on the size and location of the probe points and will be placed on the specified layer.

Analysis|Layer Compare

The Layer Compare command compares the content of two layers, and reports any differences found.

To compare two layers

1. Use the Edit|Align Layers command to align the two layers you wish to compare.
2. Select the Analysis|Layer Compare command. The Compare Layers dialog box appears.
3. Click the Add button.
4. Specify the number of the layer to compare to in the Source Layer box. If desired, you can click the arrow button to view a list of your layers, and select it from there.
5. Specify the layer to compare with the source layer in the Compare Layer box. Again, you can click the arrow button to view a layer list and select it from there
6. If two items are outside the Tolerance distance you specify, they are considered to be at different locations, and are reported as errors (differences).

7. In the Maximum Errors box, select a maximum number of errors (differences) to be displayed. This prevents the analysis from generating a huge report if you enter an incorrect tolerance.
8. Click OK.
9. To run another layer comparison on additional layers, click the Add button, and repeat steps 4-8. Add as many layer pairs to the list as desired, using the same method.

   If you need to change the parameters for a particular layer comparison, select the desired layer pair from the list and click the Edit button. To delete a layer comparison, select the desired layer pair and click the Delete button. The Empty button clears the entire list.
10. When you are ready to run the analysis, click the Run button.
11. When the analysis is finished, you are told how many differences, if any, were found. The items are listed in the Analysis list of the Navigator, under "Compare Layers" (the layer numbers are also noted). You may view the differences in the workspace, and generate a report file, using the Navigator.

Analysis|Find Duplicates

The Find Duplicates command finds all occurrences of duplicate data. For an item to be considered a duplicate, it must be at the exact same location, on the same layer, and have the same properties.

To scan for duplicate data

1. Select the Analysis|Find Duplicates command. The Find Duplicates dialog box appears.
2. Click the arrow button to view a list of layers in your design.
3. In the Select Layers dialog box, select the layers you wish to check for duplicate data, and click OK.
4. Click OK in the Find Duplicates dialog box. The analysis is run and you are told how many duplicate items, if any, were found. The items are listed in the Analysis list of the Navigator, under "Duplicate Items." You may view the errors in the workspace, and automatically fix them, using the Navigator.

Analysis|Copper Area

Toolbar Button: 

Copper area analysis calculates the amount of copper used on a layer, using a high resolution bitmap method.

To calculate copper area

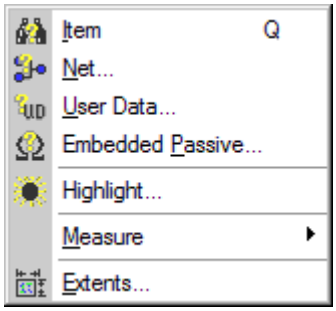
1. Select the Analysis|Copper Area command. The Copper Area dialog box appears.
2. By default, all visible layers are scanned. To specify different layers to scan, click the browse button next to the Layers box, and select the desired layers from the Select Layers dialog box.
3. Select By Window if you wish to calculate the copper area of a specific region that you define with a window. Otherwise, all visible data will be scanned.
4. Select Use Drill Information if you wish to have drill information used in the copper area calculation. If you do, then you must also specify the NC layer that contains the drill data you wish to use, and the thickness of the board.
5. Click the OK button. The copper area is calculated and a report is displayed in a text editor. The report header provides file, date, and time information. You are given the units of measure, and the body of the report is broken down on a layer-by-layer basis. The total copper area is given at the end of the report. You can save or print the report, as needed.

Related topics

Units and Precision

Query Menu

The Query menu commands provide information about the loaded database.



Query|Item

Toolbar Button:

Equivalent Hotkey: Q

The Query|Item command displays information about individual items within the database, using the Selection Filter as an aid in selecting the desired type of item.

If more than one item exists in the location you select, the Choose Selection dialog box appears. As you click on items in the Choose Selection dialog box, each selected item is highlighted, and its attributes are displayed in the Item Properties box.

⚠ *Polygon Voids imported from CAM350, ODB++, or 274-X files are supported, but are not recognized as standalone entities. Voids are only relative to the closed and filled polygon that contain them. If you query a polygon void, the information returned is for the polygon itself. Polygon voids also cannot be modified, added, or deleted.*

Related topics

Item Properties Display

Query|Net

Toolbar Button:

The Query|Net command allows you to highlight nets in a variety of colors. You may select nets by pointing and clicking, by net number, or by its UserData. The User Data query is especially useful after importing an IPC-D-356/A netlist, because the IPC net name and component pin information is saved as UserData after importing.

To highlight a net

1. Select the Query|Net command.

   If no netlist is present, you are asked if you wish to create one. Click the Yes button to automatically launch the Tools|Netlist|Generate command.

2. The Query Net dialog box appears, which controls how your nets are selected and highlighted.

   Query By Mouse Or
   By default, you may select a net at anytime by clicking anywhere on a line segment or flash. The Selection Filter is available to aid you in this. If multiple items with more than one net is in the selection area, a Choose Selection dialog box appears, so that you can pick the correct item. You may also search for nets by their VisualCAM Net number or their User Data by selecting either the **Net #** or **User Data** option, and entering the desired value in the **Search For** box and clicking the **Find** button.

   Net Information
   The information box underneath the Search For text box displays information about the last selected net. The information includes the number of pads and traces on the net, the length of the net and an approximation of its area.

   Add User Data
   Once you have a net selected, you may globally add a User Data value to all items in the selected net. This provides an easy way of assigning meaningful net names to your nets.

   Color
   By default, nets are highlighted in white. You may click on the Color square to select a new color for your next highlighted net. The color of previous selections is not altered.

Clear Highlights
Click this button to clear any data currently highlighted. This does not change any information; only the highlighting is cleared.

Zoom To Net
Select this option if you wish to have queried nets displayed in their entirety on the screen.

3. Query the nets using the desired controls in the dialog box. When you are finished, click the Close button or press the Esc key.

Related topics

Comparing Netlists Tutorial

Query|User Data

Toolbar Button: 

The Query|User Data command highlights all items that contain a specific User Data value.

To highlight items based on UserData values

1. Select the Query|User Data command. The Query User Data dialog box appears.

Search For
Enter the desired User Data value to search for in this box. The search text is not case sensitive. For a successful match, the search value can be located anywhere within an item's User Data value. This means that a search value of **DATA** would match both **DATA32** and **MYDATA**.

Color
By default, items are highlighted in white. You may click on the Color square to select a new color for your next highlighted item. The color of previous selections is not altered.

Clear Highlights
Click this button to clear any data currently highlighted. This does not change any information; only the highlighting is cleared.

2. After you have made your selections in the dialog box, click the Find button to find and highlight all items that contains the value in the Search For box. The information box underneath the Search For box displays information about the item being queried.
3. When you are finished, click the Close button or press the Esc key.

Query|Embedded Passive

Toolbar Button: 

This command is used to view information about any embedded passive in the currently loaded design. You can query an embedded passive either by clicking on it in the workspace, or entering its reference designator in the Query Embedded Passive dialog box.

This command functions the same as the Tools|Embedded Passives|Query command.

To query by mouse

1. Select the Query|Embedded Passive command. The Query Embedded Passive dialog box appears. Note that the Selection Filter alters so that you can only query the flashes that compose an embedded passive.
2. Click on the embedded passive in the workspace that you wish to view information about. It is highlighted in the workspace, and its reference designator automatically appears in the Search For box. The resistance value, in Ohms, as well as the X:Y location of the center of the resistor, are noted.
3. Click on the next embedded passive you wish to query, or use the First, Next, Previous, or Last buttons to cycle through all the embedded passives in the workspace.
4. Click on the Close button to end the function.

To query by reference designator

1. Select the Query|Embedded Passive command. The Query Embedded Passive dialog box appears.
2. Enter the reference designator of the desired embedded passive in the Search For box.
3. If you wish to have the workspace automatically zoom to the embedded passive, select the Zoom To Embedded Passive option.
4. Click the Find button. The embedded passive is highlighted in the workspace. The resistance value, in Ohms, as well as the X:Y location of the center of the resistor, are noted.

5. You can enter another reference designator, or use the First, Next, Previous, or Last buttons to cycle through all the embedded passives in the workspace.
6. Click on the Close button to end the function.

Related topics

Documentation|Reports|Embedded Passives

Query|Highlight

Toolbar Button: 

The Query|Highlight command highlights all occurrences of one or more specified database items, on selected layers, in the workspace. A count of the number of occurrences also appears in the Status Bar.

To highlight items in the database

1. Select the Query|Highlight command. You are prompted to specify which items to highlight by changing the Selection Filter.
2. In the Selection Filter, select the Types of items to highlight, and the Layers to look at. If you want items that are created using only a specific D-code or NC Tool, right-click on each of those categories and make your selections.

As you make your changes to the selection filter, the items that meet your criteria are highlighted in the workspace, and the exact number of occurrences is given in the Status Bar. The selected items remain highlighted until you turn off the highlight with the View|Highlights command, press the H hotkey, or you select another group of items with this command.

Related topics

Query|Item

Query|Measure

The Measure command presents menu that allows you to measure distances in three different ways: point-to-point, edge-to-edge, and center-to-center.

Query|Measure|Point To Point

Toolbar Button: 

The Query|Measure|Point To Point command provides measurements between two points that you select.

To measure between two points

1. Zoom or pan to the area you wish to measure in the workspace.
2. Select the Query|Measure|Point To Point command. You are prompted to select the base point.
3. Click on the point in the workspace you want to measure from. A line is attached to the cursor.

   -or-

   If you want to select exact coordinates as your base point, click the Abs button in the XY bar and specify the coordinates.
4. Move the cursor to one or more points you want to measure to. The actual distance between the edges of the items in X and Y, as well as true length, is displayed in the Status Bar.
5. To select a new base point to measure from, repeat steps 3 and 4.

   -or-

   If you are finished, press the Esc key twice.

Query|Measure|Edge To Edge

Toolbar Button: 

The Query|Measure|Edge To Edge command measures the minimum distance between the two items (from their closest edges).

To measure between the edges of two items

1. Zoom or pan to the items you wish to measure between in the workspace. To aid in your selection, turn off any layers that you do not need to be visible.
2. Select the Query|Measure|Edge To Edge command. You are prompted to select the first item.

A bounding box is added to the cursor. Anything that falls within this bounding box is a potential selection. You can increase the size of the box by pressing the PgUp hotkey, or decrease the size (for increased accuracy) by pressing the PgDn hotkey.

3. Use the Selection Filter, if necessary, and click on the item you want to measure from. If more than one item is at the selected location, a Choose Selection dialog box appears, where you can select your desired item.
4. Click on the item you want to measure to. The actual distance between the edges of the items in X and Y, as well as true length, is displayed in the Status Bar.
5. To measure between the first selected item and another item, press the Esc key, and select another item.

   -or-

   To measure between two different items, press the Esc key twice (watch the prompt in the Status Bar), and click on the desired items.

   -or-

   If you are finished, press the Esc key three times.

Query|Measure|Center To Center

Toolbar Button: 

The Center To Center command measures the actual distance between the centers of two items. You can select any item that is on a visible layer.

To measure between the centers of two items

1. Zoom or pan to the items you wish to measure between in the workspace. To aid in your selection, turn off any layers that you do not need to be visible.
2. Select the Query|Measure|Center To Center command. You are prompted to select the first item.

   A bounding box is added to the cursor. Anything that falls within this bounding box is a potential selection. You can increase the size of the box by pressing the PgUp hotkey, or decrease the size (for increased accuracy) by pressing the PgDn hotkey.
3. Use the Selection Filter, if necessary, and click on the item you want to measure from. If more than one item is at the selected location, a Choose Selection dialog box appears, where you can select your desired item.
4. Click on the item you want to measure to. The actual distance between the center of the items in X and Y, as well as true length, is displayed in the Status Bar.
5. To measure between the first selected item and another item, press the Esc key, and select another item.

   -or-

   To measure between two different items, press the Esc key twice (watch the prompt in the Status Bar), and click on the desired items.

   -or-

   If you are finished, press the Esc key three times.

Query|Extents

Toolbar Button: 

Use the Query|Extents command to view the data extents of all loaded layers. In addition to displaying the extents information, VisualCAM also updates its internal data extent information. This allows the View|All (Fit) command to correctly center the data after you have made edits to the database.

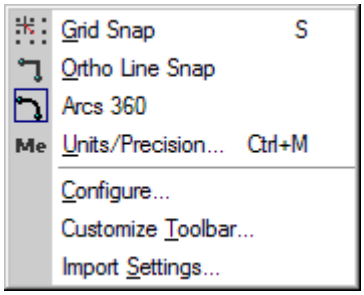
Layer
The list box displays the extents of all visible layers. The **Min** column gives the X:Y coordinates of the lower-left most extents, and the **Max** column gives the X:Y coordinates of the upper-right most extents.

True Size
If True Size is selected, the extents displayed take into account the actual size of the apertures used for each object. If it is not selected, the extents are based on the center point of each object.

Include Virtual Panelization
If Include Virtual Panelization is selected, the extents displayed include virtual panels in the calculations. If it is not selected, only the extents of the actual objects are displayed. If no virtual panels are present in the database, this option is unavailable.

Options Menu

The Options menu commands control the VisualCAM environment.



Options|Grid Snap

Toolbar Button:

Equivalent Hotkey: S

Use this command to toggle grid snapping on/off. When grid snap mode is enabled, your cross-hair cursor will automatically jump to the nearest grid point as you move it across the workspace. See the Options|Configure command for information on changing the appearance of the grid.

Options|Ortho Line Snap

Toolbar Button:

This command allows you to toggle orthogonal snap mode on/off. When enabled, all lines drawn interactively will be forced to the specified angle. See the Options|Configure command for information on changing the snap angle.

Tip: The current setting may be temporarily overridden by holding down the Ctrl key.

Options|Arcs 360

Toolbar Button:

This command allows you to toggle interpolated arcs on/off. This setting affects the method of creating arcs used by the Add|Arc Ctr and Add|Arc 3 Pt commands. If enabled, all arcs will be created using 360° circular interpolation. If disabled, all arcs will be created using small line segments. This does *not* affect the way Gerber data are imported; it only pertains to adding new arcs with the above-mentioned commands.

Warning: Not all photoplotters support circular interpolation.

Options|Units/Precision

Toolbar Button: Me

Equivalent Hotkey: Ctrl+M

This command serves as a shortcut to the Options|Configure/Units and Precision dialog box.

Options|Configure

This command allows you to change the way VisualCAM is currently configured and many of the default settings that VisualCAM uses at startup. The Configuration dialog box contains the following categories, which can be accessed by clicking on their respective tabs:

General

This tab displays various general program settings.

Ignore Files With These Extensions
These are the file name extensions that you do not use for any of the file formats that the Import Wizard handles (Gerber, aperture list, NC drill/mill, HPGL, Barco, DXF, or tool list files), but that may appear in the same folder as your databases (text files, ACR files, etc.). When the Import Wizard scans the files in the folder you select, it will skip files with these extensions. The more extensions it is able to ignore, the faster the Import Wizard will be able to scan the specified folder.

Suppress Import Warning Messages
If you wish to prevent warning messages from being displayed when importing files, select this option. A log file will still be created, but you will not be prompted whether to view it.

Print Border Text
These are the variables for border text you wish to use on plots. The keywords are:

$TIME displays the time.

$DATE displays the date.

$DESIGN displays the .vcam file name.

$LAYERNAME displays the layer name

$PROG displays the "VisualCAM" program name.

Text Editor
This is the executable file for the text editor VisualCAM will launch when you are presented with a file to view or edit.

Max Layers
This sets the default number of layers that VisualCAM creates on startup (as shown in the Color Bar and Layer Table). The valid range for this value is 32-999. Use the minimum value that satisfies your requirements to conserve memory. This option is not a layer limit. You open/import more than this number of layers, add VisualCAM will display them accordingly.

Tip: When you change this parameter, it will not come into effect until the next time you start VisualCAM.

Chord Angle
The chord angle used when creating segmented arcs. For example, a chord angle of 5° would result in a 18 separate line segments for a 90° arc.

Line Snap Angle
The angle at which lines will be forced to, if Ortho Line Snap is enabled.

Flags
This allows you to control some aspects of VisualCAM's low-level operations. Typically you would be instructed by VisualCAM Technical Support personnel on how to modify this parameter. The value is entered as a hexadecimal number.

File MRU Size
Controls how many files appear in the Most Recently Used list in the File menu.

Enable Undo/Redo
This controls whether the saving of Undo information is on or off, and whether it should be initially on upon startup. If Undo is currently enabled, and you disable it, any current Undo information will be destroyed and Undo will then be disabled. This should only be disabled if you do not want to be able to Undo any edits you perform, and want to free any memory that might be used by the Undo buffer.

Tip: The current Undo status is shown in the Status Bar.

Display

This tab displays the current settings that affect the VisualCAM desktop.

Grid
You may select a pre-defined grid size or, by entering a value in the size X:Y fields, you may specify a non-standard grid size.

Film Box
You may change the current film box size, by editing the size X:Y fields.

The default color of the film box is yellow. However, if you open a .vcam file that has a different film box color specified, it will override the default setting. That new color will become the default for new databases unless you change it. You can change the film box color by clicking on the Color button and selecting a new one from the Color Selector. If you wish to hide the display of the film box, change the color to the same as the background.

Zoom
The Limit field controls how far you can zoom in on data in the workspace. On certain combinations of screen resolution and file format, the display of items at extreme magnification can appear distorted. This setting allows you to prevent this from occurring.

This End Cap field specifies when VisualCAM should stop attempting to draw end caps on drawn lines. If the thickness of a line (in pixels) is less than or equal to this parameter, no end caps will be drawn. Higher values provide decreased redraw times at minimum zoom levels.

This parameter only affects redraw speed and has no affect on your database.

Highlight Colors
These buttons control the colors used when highlighting database items. The default color for all highlighting is white. To change a color for a particular type of highlighting, click on it and select a new one from the Color Selector.

Cross Hair
The X and Y fields provide control over the size of the cursor cross-hairs. Enter "0" in both fields for a full screen cursor. Select Show Coordinates if you want the coordinates of your cursor location attached to the cursor.

Display Draws As Sticks
Select this option if you wish to have lines drawn as sticks (instead of outlines) when sketch mode is on.

Minimize Redraws
Selecting this option minimizes the amount of automatic redrawing that is done in the workspace. If you are an experienced user, you may be comfortable redrawing the screen only at your request (using View|Redraw). Regardless of this setting, you may always interrupt a redraw, without affecting the current command, by pressing the Esc key.

Tip: The current Redraw status is shown in the Status Bar.

Color Bar Redraws Automatically
When this option is selected, if you change a draw or flash color on the Color Bar, or turn a layer on or off, the display in the workspace automatically updates. If you do not want the display automatically redrawn, unselect this option.

To override this option when you turn a layer on or off (on a case-by-case basis) press the Ctrl key when you click on a layer button in the Color Bar.

Background Color
To change the workspace background color, click this button. Then select a new color from the Color Selector.

Function Key/Mouse

This tab displays the current function key/mouse command assignments.

Mouse Buttons
These options control which view function is assigned to each mouse button. To change the function of a button, select a new one from the appropriate dropdown list. By default, the right mouse button opens a shortcut menu, so it is not assigned a function here. If you wish to use the mouse button for a function other than the shortcut menu, select your desired function from the list, then click on the Hot Keys tab and unselect the Right Click Popup Menu option.

Tip: If you disable the right-click shortcut menu, and leave the right mouse button function unassigned here, the right mouse button will act like the Esc key, and terminate command functions.

Function Keys
Allows you to specify what menu functions or macros are executed by the function keys on your keyboard. F1 and F10 are reserved by Windows, the others may be changed to any of the available functions that are listed. If you want to associate a particular macro with a function key, you must first load it using the Macro|Load command. Then it will be available for selection (all macros are listed together at the beginning of the dropdown list).

Ap List Converters

This tab shows all aperture list ACR Files available for use. Converters that are currently loaded are at the top of the list with a check mark. When selecting which aperture list converters should be loaded, a rule of thumb is "less is better"; you should unselect any aperture list converters that you do not require.

The Count column displays use count of each converter. To speed up the import process, the Import Wizard analyzes the most used converters first during automatic aperture list conversion. If you ever wish to restart the use count, click the Clear Use Counts button.

To change the location of your ACR files
The Source Folder box displays the folder where the aperture list converters are located. By default, this is the "Apconv" folder located in the VisualCAM program folder.

1. If you have not already done so, move all the ACR files you wish to use to a single folder.
2. Type a different folder name in the Source Folder box, or click the browse button to search for the folder.
3. The aperture converters will appear in the list, where you can select or unselect them for use, as desired.

To add additional ACR files

- If you have not already created the ACR file you wish to use, create one using VisualCAM's Aperture Converter Creator, and save it in the folder that is specified in the Source Folder box.

  -or-

  If you already have an ACR file, copy it into the folder specified by the Source Folder box.

To change the title of a converter
1. Click on the converter in the Description column.
2. Click on it a second time. An edit box appears around the name.
3. Type in the new name, and press Enter.

Paths, Files, Extensions

This tab displays various program default values regarding paths, files and extensions.

Startup Folder

This field specifies the default file folder that the system should use for commands like File|Open or File|Save. Click the Browse button to search for a folder. (Entering a folder path here does not take affect until the next time you start VisualCAM.)

Pre V9 Custom Apertures

If you use legacy custom aperture files (created before version 9), and want to specify a default directory to look for them, specify the directory in the Default Folder box. The aperture list file name for your pre-version 9 custom apertures should be specified in the Aperture List box.

Default Extensions

The default filename extensions used for the indicated file types are listed here. These settings affect the various Save and Open dialog boxes. More than one extension may be entered. The first extension within each list is used as the primary extension, and will be appended to any file names that are entered into Save dialog boxes without an extension.

User Menu

This tab displays the current User menu configuration and allows you to make changes as desired. The **Menu Label** column presents a list of the currently enabled User menu commands, by the name that appears in the menu. The **Command/Macro** column is the actual name of the macro or VisualCAM command that belongs to each menu label.

To add a command

1. If the command you wish to add is a new macro, you must first create the macro and save it in the Macros folder. Then select the Macro|Load command to load your macro file. You may then select the Options|Configure command, and click on the User Menu tab.
2. Type a name for your new menu item in the Label text box. A character prefixed with the "&" character will be considered the menu item hotkey.
3. Select your desired function from the Command list. This can be either a macro or an existing VisualCAM menu command.

   If you want to select a macro that does not appear in the list, it has probably not been loaded. Click the Cancel button to exit the Configuration dialog box, and select the Macro|Load command to load the macro file.
4. Click the Add button to add your new User menu command.

You can use the Move Up and Move Down buttons to reorder the User menu commands. To remove an item from the menu, select the item in the Menu Label list and click the Remove button. If you add an item and then want to change it, you must remove it and re-add it.

Macro Files

This tab displays a list where you select which Macro files you want to load automatically the next time you start VisualCAM (this does not run the macros, it simply makes them available for running on demand). If you want to run a macro immediately, use the Macro|Load command to load the macro, and then select Macro|Run.

Filename

This list shows all Macro files found in the Source Folder. You may select or unselect Macro files to load, as required.

Source Folder

Allows you to specify the folder where the macros are located. The default location is the VisualCAM "macros" folder. To add additional macro files, simply copy them into the folder specified by the Source Folder field.

Units and Precision

This tab displays the current units and precision settings that are used to represent sizes and distances (i.e. coordinates). Change the settings, as desired.

Warning: There is a possibility of rounding errors when you switch between different settings. Use care when making changes.

Hot Keys

Right-clicking your mouse within the workspace opens a context-sensitive shortcut menu of available hotkeys. This tab allows you to control what commands are listed, and in what order.

Right Click Popup Menu

Select (check) the Right Click Popup Menu option if you wish to have the shortcut menu appear when clicking the right mouse button. Unselect the option to not have the right-click shortcut menu appear. When the menu is disabled, the right mouse button then functions like the Esc key (cancels or exits functions), or it can be assigned to other functions (see the Function Key/Mouse tab).

Nested Command
Select each item you wish to have appear in the shortcut menu. To select all commands, click the Enable All button. The Enable All button then becomes Disable All, which will subsequently allow you to unselect all commands.

Move Up/Move Down
To move a list item up or down one position in the list, click on the item, then click the Move Up or Move Down button, as desired.

Hot Key Emulation
To aid new users, VisualCAM can emulate many of the hotkeys of other CAM systems. If you wish to have the hotkeys in VisualCAM emulate another program, make your selection here.

NC Settings

This tab displays the current dialect and display settings for your NC data.

NC Dialect
Select the dialect of the machine or controller you intend to use. You can ultimately export your NC data to any dialect, but if you preselect it here, VisualCAM will ensure that all data that you create or modify is done so in accordance with the specifications of the selected dialect.

Display Up Path

This displays the path of the retracted mill or drill tool as a thin line on the NC layer. The T hotkey and  button also toggles the tool path view.

Tip: After NC data are changed, the workspace should be refreshed (press the R hotkey or toggle the tool path view off and on again) to update the displayed path.

Display NC Group Order Number
When enabled, displays the NC group sequence numbers, as defined by the Tools|NC|Set Group Order command, in the workspace

Drill Display Settings
You can select whether to have your drill data (including individual hits, slots, drilled circles, and drilled text) displayed using the Tool color (defined in the Setup|NC Tools function) or the Layer color (defined by the flash color in the Color Bar or the Setup|Layers function).

By default, the drill hits are displayed as cross hairs. You can change the size of the cross hairs by changing the X:Y values in the Pixel Size box. If you unselect the Display Cross-hair At Each Drill Hit option, the hits will appear as filled circles in the workspace, their size determined by the diameter of the tool used to create them.

Show Mill Features
These options control the display of mill information in the workspace.

Plunge/Retract
Displays the plunge and retract locations of the milling tool.

Direction
Displays arrows pointing in the direction that the mill tool travels in a path.

Compensation
Displays all compensated mill paths in their true, compensated state (the position in which the mill path will actually be created).

Tab Details
Displays all tabs in their true appearance. If you unselect this option, tabs are displayed using a simple marker, as shown below.



Note that the size of the marker depends upon the width of your mill path; if you use a small tool, you may not be able to see the marker if your view of the workspace is zoomed out too far.

Mill Path Colors
This defines the color of mill data in the workspace. You can select whether to have your mill paths and circles displayed using the Tool color (defined in the Setup|NC Tools function) or the Layer color (defined by the draw color in the Color Bar or the Setup|Layers function). Other item markers are individually controlled here; just click on a color to change it.

Options|Customize Toolbar

This feature allows you to add or remove buttons from any toolbar.

The Customize dialog box has two tabs. The Toolbars tab is used for determining which toolbars are shown in the workspace, and their appearance. The Customize Toolbar tab controls which buttons appear in each toolbar.

To view a list of toolbars, with their associated buttons and functions, see the Toolbar Button Reference.

To view or hide a toolbar

- In the Toolbars tab, the Toolbars list box displays all available toolbars, with a check next to each that is displayed in the VisualCAM window. Click on the checkbox next to any toolbar to hide or display it, as desired.

To create a new toolbar

1. Click the New button in the Toolbars Tab.
2. After you assign a name to the toolbar, it appears in the Toolbars list box. The new toolbar also appears in the main window.

   -or-

1. In the Customize Toolbar tab, the Categories list box shows the types of buttons available. When you select a category, the buttons are displayed in the Buttons area.
2. Select a button by clicking on it.
3. While still holding down the mouse button, drag the button to an area outside of the dialog box. A new toolbar is created, with the chosen button and a system-assigned name.

To rename a created toolbar

1. If you wish to change the name of a toolbar you have created, select the desired toolbar from the Toolbars list (Toolbars Tab). It appears in the Toolbar Name text box below the list.
2. Modify the name, as desired. The new name is confirmed when you click on another toolbar in the list, or close the dialog box.

To delete a toolbar

- If you wish to delete any toolbar you have created, select it in the Toolbars list and click the Delete button. You cannot delete any of the default toolbars.

To modify toolbars

- While the Customize dialog box is open, you can move buttons between toolbars in the main window by clicking on and dragging buttons from one toolbar to another.

  -or-

- To completely remove a button, click on it and drag it out of the toolbar area.

  -or-

1. In the Customize dialog box, click on the Customize Toolbar tab. The Categories list shows the types of buttons available. The buttons appear in the Buttons area, and you can identify a button by clicking on it.
2. Add a button to a toolbar by clicking and dragging it from the dialog box to the desired toolbar in the main window.

If you have modified a toolbar by adding or removing buttons, and wish to return it to its default state, select it in the Toolbars list box (Toolbars Tab). Then click the Reset button.

To change toolbar display properties

- To help you identify a button’s function, descriptive text can be displayed whenever your mouse cursor is placed on a button. Select Show Tooltips if you want to see these descriptions, or unselect the option if you do not want to see descriptions.

  -or-

- Select Cool Look if you do not want the edges of the individual buttons displayed.

  -or-

- To increase the size of the buttons for easier viewing, select Large Buttons.

Options|Import Settings

The positioning and visibility of toolbars, as well as other settings related to the VisualCAM user interface, are controlled by settings in your computer's registry. When installing, default settings are put in your registry. When you change your toolbar positions, etc., those changes are maintained when you exit and re-open the program.

When a new version of VisualCAM is released, new features dictate that new registry settings are provided in the installation; so if you upgrade from one version to another, the settings from the previous version are not maintained in the new version.

This command allows you to import all viable registry settings from a previous version of GerbTool or VisualCAM, so that you can make your toolbar positions and other settings track as closely as possible from one version to another.

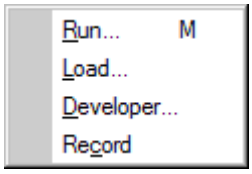
To import settings from a previous installation

1. Select the Options|Import Settings command. The Import Settings dialog box appears. It contains a list of previous GerbTool or VisualCAM installations that it has detected.
2. Select the version whose settings you wish to import, and click OK. You are asked to confirm that you wish to overwrite the current settings with the previous version.

   If you decide later that you wish to change the toolbar positions back to the default settings of the current version, use the View|Restore Toolbars command. (That command only restores the toolbar settings--no other registry settings are touched.)

Macro Menu

The Macro menu provides access to the VisualCAM macro files.



Macro|Run

Equivalent Hotkey: M

All macros loaded at program startup and through the Macro|Load command are available for execution.

To run a macro

1. Select the Macro|Run command. The Run Macro dialog box appears.
2. Select the desired macro from the Macro Name list, and click OK. The macro is executed immediately.

   -or-

1. To run a macro from the Navigator, click on the Commands tab.
2. Click on the plus (+) box next to the Macros heading to view the list of available macros.
3. Double-click on the desired macro to run it.

Macro|Load

Select the Macro|Load command to allow any macros present in a specified .mac file to be available for running in VisualCAM (using the Macro|Run command).

Tip: To automatically load a macro file at startup, see the Options|Configure command.

Macro|Developer

Selecting this menu command opens the VisualCAM Macro Developer.

Macro|Record

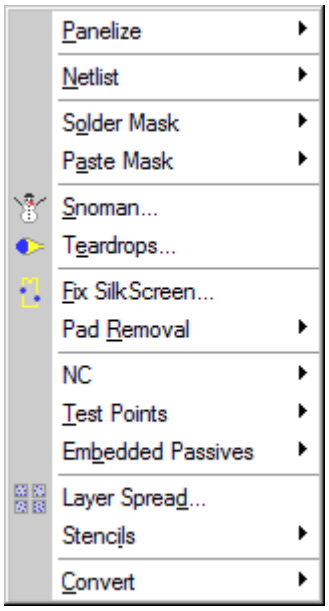
Select the Record command to record a macro by manually performing the steps in VisualCAM.

To record a macro

1. Select the Macro|Record command. The VisualCAM Macro Developer is opened in recording mode (the window is minimized, but may be expanded so that you can view your progress.
2. Perform the desired steps in your macro in VisualCAM. All your steps are noted in the Macro Developer pane that displays the current macro.
3. To stop recording a macro, select the Macro|Record command again. The Macro Developer remains open, so that you can view and modify your macro, as desired.
4. Give your macro a unique name ("MacroName" is used by default; just type over this text). This name is used to identify your macro when you load and run it.
5. Save your macro to a file.

Tools Menu

The Tools commands provide important CAM capabilities. Some commands will operate on a window or groups of items as well as complete layers.



Tools|Panelize

There are two methods of panelization. The Advanced method allows you to take one or more existing VisualCAM (.vcam) databases and define the panel either interactively or based upon a pre-existing panel template. Using Advanced Panelization, you can panelize more than one design; place coupons, pinning holes, and title blocks; and define venting, thieving, and robber bars--all in a single dialog box that displays a preview of your panel.

Simple Panelization allows you to take the currently loaded file and create multiple copies of the design for a panel, and create a simple vent pattern, if desired. Simple Panelization can be used when you want to create a one-time, basic panel of a single image. It is commonly used for creating assembly panels.

Tools|Panelize|Advanced

Toolbar Button: 

Advanced Panelization allows you to panelize single or multiple designs, and lets you store an unlimited number of panel templates so you can use the same panel configuration for multiple jobs. You must have each design already saved as a VisualCAM .vcam database (this command does not automatically operate on the design currently loaded in the workspace). Coupon placement, venting and thieving, pinning holes, and title blocks are supported, as long as those elements have been saved as individual .vcam files.

The Advanced Panelization dialog box is designed to step you through the panelization process. Each feature under each tab is defined below. For detailed procedures on creating panels, templates, and other tasks, see the *Panelization Tutorial*.

⚠ *Tip: For best results when creating a new panel, do not have a database open in the workspace. You should only open a database first if it is a panel template or if you are modifying a pre-existing panel.*

Panel

Panel Size
Define the X and Y dimensions of your panel. The default values are the film box size. The currently selected units of measure are used.

Panel Origin ⊠
Select an origin (0:0 point) for the panel. All image data will be placed in reference to it.

Tooling Origin Offset From Panel Origin
If desired, specify the X:Y offset of the tooling origin from the panel origin. This allows you to specify a separate origin for NC data (all drill and mill coordinates reference this point).

Image

You define the placement of each selected image in the Image and Layout tabs.

Image Files
All the images listed in this box will be included in the panel. The currently selected image is the active image, (the selected settings apply to the image you have selected in the list). To add an image to the panel, click the Insert button and type in the full file path, or browse for the desired .vcam file. After entering the file in the list, press the Enter key to accept it. To remove an image from the panel, select it in the Image list and click the Delete button.

Change Anchor Point From Image Origin To
The anchor point establishes an insertion point for the images. This setting is most important when you use Manual image placement or Image Rotation (under the Layout tab). By default, the image origin (0:0) is used as the anchor point. Select this option if you wish to select another location. The selections use the appropriate location on the image's data extents/border.

Border
Specify whether the image's border should be obtained from the border layer as identified in the .vcam file, or the data extents. If you use the border, specify whether you want clearance measurements to be taken from the center or the edge of the border line. If the .vcam file does not have a border layer tagged, the From Border Layer option cannot be selected.

Layer Mapping
Each panel has a layer structure, and layers from each image need to be placed on the proper panel layer. It is usually done automatically for you, if you have correctly tagged layers in your .vcam files. The Layer Mapping dialog box lists the panel layers in the Mapped Layers list, next to the , with the associated image layers underneath each of them. Any image layers that have not been associated with a panel layer appear in the Unmapped Image Layers list. They must be mapped in order to be included in the panel. To map a layer, click on it in the Unmapped list and drag it to the appropriate panel layer.

By default, the panel layers are listed in the same order as the first design loaded into the panel. The layer types are indicated, but layer names will be <empty>. If you wish to assign the layer names from the image to the panel layers, click the Copy Mapped Layer Names button. It is recommended that you assign layer names to the panel layers if you perform Venting & Thieving or add Robber Bars. The layer names are used in creating layer sets.

Warning: If you change the layer definition (add a layer, change a layer type, etc.) of an image after creating a panel, you must use the Layer Mapping dialog box to map the new/modified layer to the appropriate panel layer. It will not be done for you automatically.

When you define the layer mapping, if any NC data exists in the images, you are also required to map tool definitions to your template's tool table. If you are not working with a template, or did not define a tool table in your template, the first image you load into the template will define the tool table. If you load additional images into the table, you must then map their tools based upon the definitions from the first image.

Layout

Image Files
All the images listed in this box will be included in the panel. The settings apply to the image you have selected in the list (they change dynamically for the selected image). To add an image to the panel, click the Insert button and browse for the desired .vcam file. To remove an image from the panel, select it in the Image list and click the Delete button.

Tip: When changing any of the amounts for the below options, press the Enter key after you type in an amount.

Image Layout
These selections define the spacing and number of images on the panel.

If you choose *Minimum Spacing*, you define the minimum amount of X and Y spacing between the images, measured from their borders. VisualCAM then automatically calculates how many images can be on the panel.

If you choose *Exact Spacing*, the images are spaced exactly the amount you specify. If the resulting layout is not centered on the panel, you can have VisualCAM center the images but maintain the border-to-border spacing, by selecting Center Images Within Margins.

Number Of Copies calculates the image-to-image spacing based upon the number of images you specify in the X and Y directions.

Manual Placement allows you to specify the number of images you want in the panel, and then move and rotate the images as desired.

To move an image, right click on it and select the Move command from the shortcut menu. In the Image Location dialog box, select the new coordinates of the insertion point.

To rotate an individual image, right click on it and select the Rotate command from the shortcut menu. In the Rotate Image dialog box, select the number of degrees to rotate the image around its insertion point, counter-clockwise.

To delete an individual image, right-click on it and select the Delete command from the shortcut menu. The image is immediately removed.

Image Rotation

Rotates all the images, around their insertion point, counter-clockwise. Note that this selection will override any manual rotation of individual images (performed with the right-click shortcut menu).

Panel Edge To Image Spacing

You can select Symmetrical and specify the Minimum Spacing that the image borders must maintain from all the panel edges, or select Margin to define the offset from each individual edge.

Tip: When the panel is finished and appears in the main workspace, if you move the images, the Minimum Spacing or Margin Spacing values will be reset based on the new placement of the images.

Robber Bars

Layer Set
Robber Bars are usually unique to a layer's type. To quickly apply a specific robber bar to a group of layers, you need to create layer sets. To create a new layer set, click the Define Layer Sets button. The Panel Layer Set Setup dialog box appears. Click the New Layer Set button to create a new layer set. You are prompted for the name of the new set. Create as many layer sets as necessary. Drag and drop layers from the Layer List into the appropriate set in the Layer Set List. If you make a mistake, layers can be dragged back into the Layer List, or you can delete layer sets. The actions you perform here have no effect upon the actual panel layers (they are not deleted or reordered).

Add Robber Bar
Select this option to add a robber bar to your layer set. You must then define the appearance of the robber bar.

Width Of Robber Bar
The width of the solid robber bar "line".

Clearance From Panel Edge
The distance of the outside edge of the robber bar from the panel edge.

Generate Robber Bars On These Edges
Select the sides of the panel that you want to place the robber bar on.

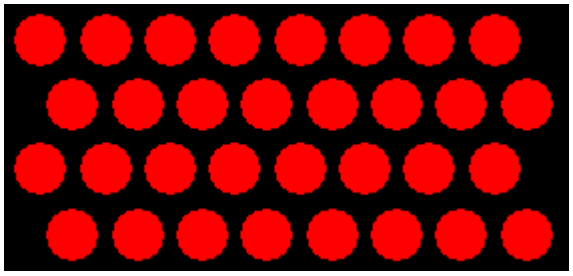
Venting & Thieving

Layer Set
Venting and thieving patterns are usually unique to a layer's type. The layer sets used for venting and thieving are the same as those used for robber bars.

Pattern
After you select the layer set you wish to work on, click on the desired pattern type (or select click on None to have no venting or thieving on the layer set). You are presented with a preview of the pattern, and the options for defining Dot, Hatch, and Star Burst patterns. If you are defining a small dot or hatch pattern, it is recommended that you select the Pattern Preview option as you make your changes. Otherwise, in the Panel Preview mode, the pattern may be too small to view.

Tip: As you define your patterns, you may notice a delay before your changes appear in the preview. Take note of the Status Indicator in the main VisualCAM window. It will flash red when the system is calculating your changes, and turn green when finished.

Dot pattern
Select the Shape and Size of your dots. Choose whether to have the offset spacing between the dots measured to the edge or center of the dots. Define the spacing between dots in the X and Y directions. To create a staggered pattern of dots, select the Stagger option and decide which row should be offset. To have them staggered like the below example, the Row 2 offset should be half the value of the Spacing Between Columns amount, and the Row 1 offset should be zero. Select the polarity of your pattern.



Hatch pattern
You can define up to 3 lines. The Size is the thickness of the line. The Spacing Between Centers is the distance between the lines, measured from their centers. The Angle is the angle of each line, in degrees. Select the polarity of your pattern.

Star Burst pattern
Select the number of spokes (lines) you want in the star burst pattern, and the desired width of each spoke as measured at the panel's edge. Define the angle of the first spoke in the panel (this is the spoke at 0:0, or 12 o'clock). Select the polarity of your pattern.

Save & Load Pattern buttons
Click the Save Pattern button to save the pattern definitions as a .pat file for future use. Click the Load Pattern button to load a previously-created pattern file.

Clearances
For all patterns, you need to specify the clearance of the venting data from the panel edge and the other data that is on the panel.

Tools|Panelize|Flatten

Executing this command immediately removes the step & repeat codes from a panelized database, and essentially creates a single design from a panel.

Tools|Panelize|Simple

Toolbar Button: 

The Simple Panelization is mainly used for creating "simple" assembly panels. If you are creating panels for production tooling or more complex assembly panels, use Advanced Panelization.

Some data preparation is necessary prior to panelization. Using the Setup|Layers command or the Navigator, make sure that each layer is "tagged" with the correct layer type. All visible layers will be included in the panel. Turn off any layers that you do not wish to include in the finished panel (such as drawing layers). Although only visible layers will be copied, all layers of the original image will remain aligned after panelization.

Remove any extraneous data on your electrical layers that are outside the perimeter of the board, such as title blocks, targets, crop marks, etc. Complete any other data preparation and analysis that is necessary prior to sending your job to manufacturing.

The finished size of your panel is determined by the size of the film box. To change the size of the film box, select the Options|Configure command, click the Display tab, and change the film box X:Y values. All your data must be contained within the film box area. Change your origin, if necessary. (It is recommended that your origin be the same as the lower-left corner of the film box.)

To panelize two or more PCBs, if you are working from Gerber data, import all the Gerber files for your databases together. If you are working from VisualCAM .vcam files, open one of the files, then merge in the second. To successfully panelize multiple PCBs, the PCBs must have the same number of layers, and each of the corresponding layers should be the same type.

The system executes the panelization process immediately after you click OK. If you selected the Auto Calculate option, the images are centered in the film box. If you did not select Auto Calculate, after clicking OK, you must window around the data you wish to panelize. Manual panelization (Auto Calculate not selected) does not center the image into the film box.

If you wish to place the vent pattern on more than one layer, it is recommended that you place the vent pattern on an empty layer, then can use the Edit|Copy command after panelization to copy it to the desired layers. Placing the pattern on an empty layer makes it easier to copy. The Add|Array command also allows you to fill an area with a vent pattern, however, it does not offer a clearance around the images.

Following is a description of each option in the Panelize dialog box.

Standard
If this button is selected, VisualCAM panelizes your data using copies (not step & repeat codes). This results in a bigger database size, but it allows you to manipulate the individual copies after panelization is completed.

Virtual
Clicking the Virtual button in the Panelize dialog box allows VisualCAM to panelize your design without actually duplicating layer data (it uses step & repeat codes). The benefits of virtual panelization include automatic updating of all images, if you alter the "master" image, and drastically reduced file sizes. However, you are prevented from manipulating the individual panelized images.

The display of "virtual panels" may be toggled on/off using the Ctrl+V hotkey.

Step & Repeat Codes
If your designs are to be plotted on a 274-X or FIRE9XXX compatible plotter, VisualCAM will automatically insert the proper step & repeat codes into your Gerber data. Although no data are duplicated during virtual panelization, the data origin is modified to center the images within the panel. Therefore, it is still necessary to save your design after panelization.

VisualCAM will also insert step & repeat codes into NC data, if the Virtual button is enabled. This may be useful in drilling large panels, if your NC equipment has limited memory.

Current Panel
Virtual panelization requires that you setup layer sets. The names assigned to the existing virtual layer sets appear in the Current Panel list. Select the set of layers that you wish to include in the panel from the list.

Setup Virtual Panels
If you wish to create a new layer set, click the Setup Virtual Panels button, and select the Add command. In the dialog box, type a name to your layer set, and click OK. Then, in the Layer Set Setup For Virtual Panels dialog box, drag the desired layers for the set from the Layer List to the Layer Set List. Click OK when you are done. You may now create a panel from this layer set.

If you wish to delete a layer set (whether it is inactive or a virtual panel that you have already created), select the desired panel from the Current Panel list. Click the Setup Virtual Panels button, and select the Delete command. If the layer set was used in a virtual panel, the step and repeat images from the virtual panel are deleted when you click OK in the Panelize dialog box.

If you wish to modify a layer set (whether it is inactive or a virtual panel that you have already created), select the desired panel from the Current Panel list. Click the Setup Virtual Panels button, and select the Modify command. The Layer Set Setup For Virtual Panels dialog box appears, where you can change the layers that are in the set. If the layer set was used in a virtual panel, the step and repeat images are changed when you click OK in the Panelize dialog box.

Auto Calculate...
Select this option if you want the system to determine how many copies to place on the panel, based on values you enter for spacing between images and from the film box. If you want to perform "manual panelization," and specify the number of copies on the panel and the point-to-point placement of each image, do not select this option.

If you select Auto Calculate, the system executes the panelization process immediately after you click OK. The images are centered in the film box. If you choose to perform "Manual Panelization," you must window around the data you wish to panelize, after clicking OK. Manual panelization does not center the image into the film box.

Minimum Spacing Between Images
In automatic mode, these fields specify the edge to edge spacing between the resulting panels.

Minimum Spacing To Film Box
These values are used when performing an automatic panelization. They specify the minimum distance allowed between the edge of the panelized items and the film box.

Copies
These fields allow you to specify the number of copies of the design you wish to place when in manual mode, and the number of copies the system places in automatic mode.

Tip: The number of copies in each direction must be at least one as the original image is included in the count. If you wish a single row of boards for instance, you would set the ***Y*** *value to 1.*

Image Offsets
Select the X:Y offset for each copy. This distance is the spacing from a point on one copy of the board to the same point on the next copy. The amount is added to the X:Y coordinates of each copy, to determine its placement relative to the other images. Note that if you select a value that is too small, the copies will overlap.

Auto Vent
When this button is selected, VisualCAM will automatically apply venting to the panelized board. VisualCAM allows for simple venting pattern creation, using D-codes.

Vent To Image Spacing
This value is the edge-to-edge distance between the panelized images and the venting pattern.

Pattern Spacing
These values allow you to specify the spacing between the flashes in the venting pattern.

Destination
This allows you to specify the layer that the venting pattern is placed on, and the D-code used to create the pattern. The style of vent pattern is easily customized using custom apertures. For example, you could create a hatch or cross-

hatch pattern using a diagonal or cross-shaped custom aperture. Just be sure to set the height and width of the overall size of the custom aperture in the aperture list.

Preview
Click this button to preview your panel in the workspace. You can return to the dialog box after previewing, and modify the panel definition. The panel is not finished until you click the OK button.

Tools|Netlist

The Netlist commands allow you to create and save a netlist.

Tools|Netlist|Generate

Toolbar Button: 

The Generate command processes all conductive layers and creates a netlist that becomes part of the internal database. The netlist may then be used by other functions that require a netlist, such as Netlist Compare and Design Rule Checking.

To generate a netlist

1. Using the Setup|Layers command, make sure that each layer is tagged with the correct layer type. This is required so that the system understands what each layer is.
2. Remove any extraneous data on your electrical layers that are outside the perimeter of the board, such as title blocks, targets, crop marks, etc.

Tip: We recommend that any documentation for the board be done on a separate Drawing layer. If you wish to keep the data, placing it on a drawing layer assures it will not interfere with any processing.

3. Select one of the Tools|Convert|Drawn Pads commands to convert any drawn pads to flashes.
4. Use the Analysis|DRC/MRC command to make sure your layers are properly aligned. If any layers are misaligned, you can use the Edit|Align Layers command to align them.
5. Use the Tools|Pad Removal|Stacked command to remove any redundant pads.
6. If you wish to exclude any data from the netlist generation process, use the Edit|Select|New Group command to create a Select Group.
7. Select the Tools|Netlist|Generate command. The Netlist Generation dialog box appears.
8. By default, the system will generate a "Conventional" netlist, and assume you do not use any MCM/LTCC technology, or Blind/Buried vias. If you are using MCM/LTCC technology, click the Setup MCM/LTCC Stackup button. This opens the MCM/LTCC Setup dialog box, where you define your layer stackup. If you have Blind/Buried vias, click the Setup Blind/Buried Via Layer Sets button. This opens the Layer Set Setup For Blind/Buried Vias dialog box, where you define the pairing of NC layers to circuit layers by creating layer sets.
9. An NC layer with through-hole drill information is necessary to determine layer to layer connectivity, as well as direct connections to plane layers. The system will automatically display your NC layer in the Thru-hole Drill Layer box. If you want to select a different NC layer, click the arrow button and select the layer. If you do not supply an NC layer (select "None"), the netlist extract function will use the Aperture Table Type field to determine whether each pad is SMT or not.
10. Select the Include Single Point Nets option if you wish to designate all isolated pads, pins, or vias as nets
11. If you created a Select Group of times to ignore during netlist generation, select the Don't Include Items In Select Group option.
12. If you previously generated a netlist, and need to recreate it after modifying some data, you can maintain your existing net name assignments by selecting the Preserve Existing Net Names option.
13. Click the OK button to generate the netlist. While the system is processing your date, take note of the Status Indicator in the bottom-right corner of in the main VisualCAM window. It will flash red when the system is working, and turn green when finished.

Tip: Netlist information is displayed in the Navigator. To save processing time when you generate a netlist, the Nets list in the Navigator is not populated until you first attempt to access it.

When you save your database to a .vcam file, the netlist information is saved with it. You can use the Tools|Netlist|Save command to save the netlist as a simple ASCII file, export the netlist as an IPC-D-356/A file, or export within Gerber files. To export a netlist within Gerber files, select the Netlist option in the Gerber Export Data Format dialog box. VisualCAM uses the G04 command to embed the netlist within the Gerber files. This will cause the Gerber file to increase slightly in size. It is recommended you do not include a netlist when submitting your files to be plotted, due to their increased size and the possibility that the photoplotter may not properly recognize the G04 command.

Related topics

Comparing Netlists Tutorial

Tools|Netlist|Save

This command will generate an ASCII netlist file consisting of pad X:Y coordinates. One netlist is created for all viewed layers. If you wish to export the netlist as an IPC-D-356/A file, use the File|Export|IPC-D-356 command.

To save a netlist

1. Use the Tools|Netlist|Generate command to create a netlist.
2. Select the Tools|Netlist|Save command. The Netlist Save dialog box appears.
3. In the Filename box, specify a name for your file. By default, the file is saved in VisualCAM's Samples folder. Click the browse button to save it to a different folder.
4. Specify the M.N format of the coordinates in the generated netlist file.
5. Select the Metric option to export the coordinates in millimeters. Otherwise, it will be in inches.
6. Select the Renumber Sequentially option to renumber the net numbers, if necessary, to make sure that they are output in order and with no gaps in the net numbers.
7. Choose how to label the nets. If your nets have User Data assigned to them you may choose to have your netlist labeled with the User Data instead of net numbers.

Related topics

Sample VisualCAM Netlist (.nl) File

Tools|Netlist|Clear Netlist

The Clear Netlist command removes all netlist information from the currently loaded database. After selecting the command, you are asked to confirm that you wish to remove the netlist information. This function cannot be reversed by the Undo command.

Tools|Netlist|Apply External Net Names

After you have performed a Netlist Comparison, and have corrected any errors, you can use the Apply External Net Names command to apply the net names from the IPC-D-356/A netlist to the VisualCAM netlist information. The most important purpose for this is that, when you view the net information in the Navigator or query a net, you will see the net name rather than the arbitrary VisualCAM net number. If you save your data as a .vcam file, the netlist information is saved as well, so you will no longer need to refer to the external netlist file when viewing your data.

Tools|Netlist|Set Netlist Information

This function is used to define net types (signal or power) for in-circuit test, after you have reverse engineered your design for assembly. It is only necessary to set this netlist information if you are exporting to the FATF format, and are using the file for in-circuit test.

To set netlist information

1. After you have reverse engineered your data for assembly, select the Tools|Netlist|Set Netlist Information command. The Set Netlist Information dialog box appears.
2. Each net is listed by name, and set to "Signal" by default. To change a net to "Power", click on the Type for that net, and select Power from the list that appears.
3. When you are finished setting your netlist information, click the OK button.

Tools|Solder Mask

The Solder Mask commands are used to create solder mask layers. They use a comprehensive set of rules that are designed to give you maximum possible clearance while maintaining optimal coverage throughout your design. Special support for fine-pitch pads is provided, with automatic generation of block openings for those areas to prevent web (sliver) problems between pads.

Tools|Solder Mask|Generate

The Solder Mask Generator automatically produces accurate and usable solder mask layers. It provides the maximum coverage to avoid potential exposure problems while, at the same time, providing the maximum clearance.

To generate a solder mask

1. Make sure you have tagged your layers.
2. Use the Tools|Convert|Drawn Pads|Select Draws command to convert your drawn pads to flashes.

3. If you want any items in your database ignored during the mask generation process, create a Select Group of those items.
4. Select the Tools|Solder Mask|Generate command. The Generate Solder Mask dialog box appears.
5. Select whether to create a Top Mask, Bottom Mask, or both.
6. In the Maximum Errors box, specify the number of allowable errors. The mask generator first creates the mask using your oversize parameters, and then analyzes the mask to make sure all openings abide by your minimum spacing values. If it reaches this error count limit, it stops logging errors and warns you that the limit was reached. This limit is provided in case you enter oversize and spacing parameters that are not appropriate for your design.
7. Specify your Desired Oversize. The generation process adds an opening of the appropriate shape on the mask layer at the same location as each pad on the corresponding electrical layer. The openings are increased in size by the specified Desired Oversize amount (and will not be larger than this). In case the system cannot create a mask opening as big as the desired size, specify a Minimum size for the opening.
8. Specify the Mask Opening spacing, which is the minimum spacing allowed between two openings on the mask layer.
9. Specify a Mask To Trace spacing, which is the minimum spacing allowed between a mask opening and traces on the corresponding electrical layer. If the system cannot create an opening without violating these spacing amounts, an error will be logged.
10. Select the Detect Fine Pitch Pads option to have the system analyze rectangles, oblongs, ellipses, bullets, squares, and customs for fine pitch conditions. If three or more pads of the same shape, size and angle lie in line with each other, and are within the maximum Edge-To-Edge or Center-To-Center Spacing you specify, they are considered a group. Each group of fine pitch pads is processed as one element, and a single opening is created around all of them.
11. If you want pads that are less than a particular size excluded from the process, select Ignore Pads On Electrical Layer Less Than and select a minimum pad size.
12. If you have built a Select Group of items you want to ignore during solder mask generation, select Don't Check Items In Select Group.
13. Select Fix Errors By if you want the system to automatically fix mask spacing errors. You can have errors fixed by shrinking the entire mask opening (Resizing Opening); or have the opening decreased (Shaved) on the offending side, creating an irregular shape. If the fixed opening violates the minimum oversize specified, you will be asked if the fix should be applied.
14. Click OK to generate the solder mask. Regardless of whether you have the system fix any errors, they are all logged in the Navigator, where you can view them, print reports, and fix/unfix the errors. The following possible errors are reported:

    Mask Opening Spacing Error
    Two mask openings violate the minimum distance between mask openings. This might create mask slivers.

    Mask Opening To Trace Error
    The mask opening is too close to a trace and violates the minimum mask to trace distance specified. The trace might be exposed in the mask opening.

Tools|Solder Mask|Optimize

This function analyzes your existing solder mask layers, and determines if the mask openings and spacing abide by rules that you specify.

To optimize a solder mask

1. Make sure you have tagged your layers.
2. Use the Tools|Convert|Drawn Pads|Select Draws command to convert your drawn pads to flashes.
3. If you want any items in your database ignored during the mask generation process, create a Select Group of those items.
4. Select the Tools|Solder Mask|Optimize command. The Optimize Solder Mask dialog box appears.
5. Select whether to optimize the Top Mask, Bottom Mask, or both.
6. In the Maximum Errors box, specify the number of allowable errors. If the optimizer reaches this error count limit, it stops logging errors and warns you that the limit was reached. This limit is provided in case you enter oversize and spacing parameters that are not appropriate for your design.
7. Select your Desired and Minimum Oversizes. The system looks at all openings on the mask layer at the same location as each pad on the corresponding electrical layer. It makes sure there are no openings smaller than the Minimum size. If an existing opening is larger than the Minimum size, then it is not flagged as an error unless it

violates the Minimum Spacing rule. If there is a missing mask opening, and you have VisualCAM fix it, the Desired size is used to create one. If the system cannot create a mask opening as big as the Desired size, it will create one no smaller than the Minimum.

8. Specify the Mask Opening, which is the minimum spacing allowed between two openings on the mask layer.
9. Specify the Mask To Trace spacing, which is the minimum spacing allowed between a mask opening and traces on the corresponding electrical layer.
10. Specify the Location alignment, which is the maximum misalignment allowed between the location of the mask opening and the corresponding pad on the electrical layer.
11. Select Detect Fine Pitch Pads to have the system analyze rectangles, oblongs, ellipses, bullets, squares, and customs for fine pitch conditions. If three or more pads of the same shape, size and angle lie in line with each other, and are within the maximum Edge-To-Edge or Center-To-Center Spacing you specify, they are considered a group. Each group of fine pitch pads is processed as one element, and should have a single opening around them. The optimizer will make sure this is the case.
12. If you want pads that are less than a particular size excluded from the process, select Ignore Pads On Electrical Layer Less Than and select a minimum pad size.
13. If you have built a Select Group of items you want to ignore during solder mask optimization, select Don't Check Items In Select Group.
14. If your current mask layer openings were created using drawn data, and you wish to convert the drawn data to flashes, select the Convert Drawn Mask Layer option.
15. Select Fix Errors By if you want the system to automatically fix mask errors. You can have spacing errors fixed by shrinking the entire mask opening (Resizing Opening); or have the opening decreased (Shaved) on the offending side, creating an irregular shape. Other types of errors are fixed appropriately (see below). If the fixed opening violates the minimum oversize specified, you will be asked if the fix should be applied.
16. Click OK to run the optimizer. Regardless of whether you have the system fix any errors, they are all logged in the Navigator, where you can view them, print reports, and fix/unfix the errors. The following possible errors are reported:

    Missing Electrical Pad
    There is no pad on the electrical layer corresponding to an opening on the mask layer. To fix this, the mask opening is deleted.

    Missing Mask Opening
    There is no opening on the mask layer corresponding to a pad on the electrical layer. To fix this, a mask opening is added.

    Misalignment Error
    The opening on the mask layer and its corresponding pad are misaligned by a value greater than the Location Alignment value you specified. To fix this, the mask opening is moved to the same location as the pad.

    Mask Opening Minimum Oversize Error
    The mask opening is not oversized by at least the minimum oversize specified. To fix this, the mask opening is enlarged to the minimum oversize amount.

    Mask Opening Spacing Error
    Two mask openings violate the minimum distance between mask openings. This might create mask slivers. To fix this, the openings are shaved or reduced (as specified by you).

    Mask Opening To Trace Error
    The mask opening is too close to a trace and violates the minimum mask to trace distance specified. The trace might be exposed in the mask opening. To fix this, the openings are shaved or reduced (as specified by you).

Tools|Paste Mask

The Paste Mask commands are used for creating or optimizing paste mask layers. They are driven by a comprehensive set of rules that are designed to give you optimal reduction across a wide variety of pad shapes and sizes. Special support for fine-pitch pads allows for independent X and Y reductions to eliminate paste build-up between pads.

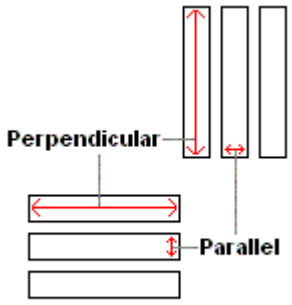
Tools|Paste Mask|Generate

The Paste Mask Generator produces accurate and usable paste mask layers. It uses the size of your conductive pads, and an undersize amount that you define, to create a paste mask layer with openings of an appropriate size.

To create a paste mask

1. Make sure you have tagged your layers.

2. Use the Tools|Convert|Drawn Pads|Select Draws command to convert your drawn pads to flashes.
3. If you want any items in your database ignored during the mask generation process, create a Select Group of those items.
4. Select the Tools|Paste Mask|Generate command. The Generate Paste Mask dialog box appears.
5. Select whether to create a Top Mask, Bottom Mask, or both. The system uses the Thru-hole Drill Layer specified to identify the different types of pads in your design. It will process any pad that is SMT. If you do not have a thru-hole drill layer, then the system will use the Aperture Table Type information.
6. In the Maximum Errors box, specify the maximum number of allowable errors. The mask generator first creates the mask using your undersize parameters. If an opening cannot be created, an error is logged. If it reaches this error count limit, it stops logging errors and warns you that the limit was reached. This limit is provided in case you enter an undersize amount that is not appropriate for your design.
7. Specify your Desired Undersize. The generation process adds an opening of the appropriate shape on the mask layer at the same location as each pad on the corresponding electrical layer. The openings are decreased in size by the specified Desired Undersize amount.
8. If you select the Round Corners option for the openings, define the corners either as a percentage of the shortest side of the aperture (1 to 50 percent), or as an absolute value.
9. Select the Detect Fine Pitch Pads option if you want the system to analyze pads for fine pitch conditions (round pads are ignored). If three or more pads of the same shape, size and angle lie in line with each other, and are within the maximum Edge-To-Edge or Center-To-Center Spacing you specify, they are considered a group of fine pitch pads. You then specify the amount of undersize applied to the perpendicular and parallel sides of each pad in each group.



Fine Pitch Pad Undersizing

10. If you want pads that are less than a particular size excluded from the process, select the Ignore Pads On Electrical Layers Less Than option and select a minimum pad size.
11. If you have built a Select Group of items you want to ignore during paste mask creation, select Don't Check Items In Select Group.

⚠ *Tip: The Fix Errors option is only available for the Optimizer. The only error that can occur while generating a paste mask layer is if the undersize amount is too large for a pad, which would mean that the opening size would effectively be less than zero. An opening would not be created, and a Missing Mask Opening error would be logged. To fix this, you must create the paste mask again, decreasing the undersize amount or increasing the size of small pads to exclude (whichever is appropriate in your case).*

12. After you have made your selections, click OK to execute the function.

Related topics

Tools|Stencils

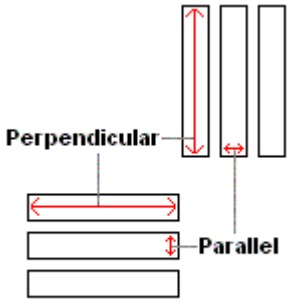
Tools|Paste Mask|Optimize

This function analyzes your existing paste mask layers, and determines if the mask openings abide by rules that you specify.

To optimize a paste mask

1. Make sure you have tagged your layers.
2. Use the Tools|Convert|Drawn Pads|Select Draws command to convert your drawn pads to flashes.
3. If you want any items in your database ignored during the mask optimization process, create a Select Group of those items.
4. Select the Tools|Paste Mask|Optimize command. The Optimize Paste Mask dialog box appears.

5. Select whether to optimize the Top Mask, Bottom Mask, or both. The system uses the Thru-hole Drill Layer specified to identify the different types of pads in your design. It will process any pad that is SMT. If you do not have a thru-hole drill layer, then the system will use the Aperture Table Type information.
6. In the Maximum Errors box, specify the maximum number of allowable errors. If the optimizer reaches this error count limit, it stops logging errors and warns you that the limit was reached. This limit is provided in case you enter an undersize amount that is not appropriate for your design.
7. Specify your Desired Undersize. The optimization process looks at all openings on the mask layer at the same location as each pad on the corresponding electrical layer. It determines if the openings are undersized by at least the minimum Desired amount. (It will not report an error if the mask opening is undersized more than the minimum amount.)
8. If you select the Round Corners option for the openings, define the corners either as a percentage of the shortest side of the aperture (1 to 50 percent), or as an absolute value.
9. The Location Alignment specifies the maximum misalignment allowed between the location of the mask opening and the corresponding pad on the electrical layer.
10. Select the Detect Fine Pitch Pads option to have the system analyze pads for fine pitch conditions (round pads are ignored). If three or more pads of the same shape, size and angle lie in line with each other, and are within the maximum Edge-To-Edge or Center-To-Center Spacing you specify, they are considered a group of fine pitch pads. You then specify the amount of undersize that should applied to the perpendicular and parallel sides of each pad in each group.



Fine Pitch Pad Undersizing

11. If you want pads that are less than a particular size excluded from the process, select the Ignore Pads On Electrical Layers Less Than option and select a minimum pad size.
12. If you have built a Select Group of items you want to ignore during paste mask optimization, select Don't Check Items In Select Group.
13. If your current mask layer openings were created using drawn data, and you wish to convert the drawn data to flashes, select the Convert Drawn Mask Layer option.
14. Select the Fix Errors option if you want the system to automatically fix mask errors. Regardless of whether you have the system fix any errors, they are all logged in the Navigator, where you can view them, print reports, and fix/unfix the errors. The following possible errors are reported:
    - **Missing Electrical Pad** There is no pad on the electrical layer corresponding to an opening on the mask layer. To fix this, the mask opening is deleted.
    - **Missing Mask Opening** There is no opening on the mask layer corresponding to a pad on the electrical layer. To fix this, a mask opening is added.
    - **Misalignment Error** The opening on the mask layer and its corresponding pad are misaligned by a value greater than the Location Alignment value you specified. To fix this, the mask opening is moved to the same location as the pad.
    - **Mask Opening Minimum Undersize Error** The mask opening is not undersized by at least the minimum specified undersize. To fix this, the mask opening is undersized by the specified amount.
15. After making your selections, click OK to execute the optimization.

Related topics

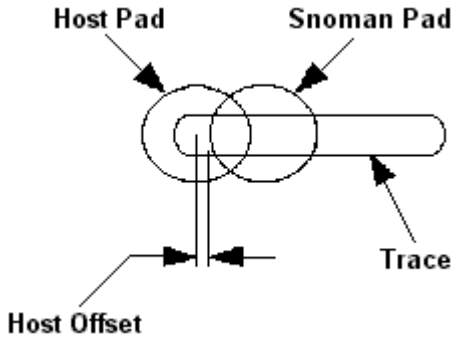
Tools|Stencils

Tools|Snoman

Toolbar Button: 

Snoman is a tool designed to create a *maximum material condition* at the point where a trace segment enters a pad, thereby eliminating the possibility of pad/trace separation (breakout). This is accomplished by examining a layer and outputting pad flashes of the correct size at the correct locations. Automatic adjustments are made to the size and location of the generated Snoman pads to eliminate design rule spacing violations. Snoman is primarily designed for working with very small pads and traces, such as micro vias that are 30 mils or less, but can be used anywhere to prevent pad/trace separation.

Snoman derives its unusual name from the appearance of a Snoman pad placed on top of a host pad, which resembles a snowman. The following illustration shows the original pad and trace as well as the resulting Snoman pad.



The distance maintained between the host pad center and the edge of the generated Snoman pad (see Host Offset in the illustration above) is adjustable. Negative values allow the Snoman pads to closely hug the host pads, if desired.

To create Snoman pads

1. A netlist is required for the Snoman tool to work properly. If one does not exist, use the Tools|Netlist|Generate command to create a netlist.
2. Select the Tools|Snoman command. The Snoman dialog box appears.
3. In the Report File box, specify the filename of a report file to log any errors. By default, the file is saved in the Samples folder. Click the Browse button to search for a different folder.
4. In the From Layer box, specify the source layer that will be processed by the Snoman command. Click the arrow button to select the desired electrical layers. If "All" is entered, all electrical layers will be processed.
5. In the To Layer box, specify the layer to place the generated pads. Click the arrow button to select the desired layer. If "Same" is entered, the source layer will be used.
6. The D-code option allows you to process flashes that only use particular D-codes. Click the arrow button to select the desired D-codes. If "All" is entered, all D-codes are processed.
7. To process only pads that fall within a selection window, select the Window option. Otherwise, the entire layer is processed.
8. A Snoman pad will not be added if it violates either of the Minimum Spacing values. Pad/Trace is the minimum spacing that must be maintained between pads and adjacent traces, and Pad/Pad is the minimum spacing that must be maintained between adjacent pads. If a violation is detected, the system will reduce the size of the Snoman pad until the violation no longer exists or the Min Percent value has been reached.
9. Select a Min Percent value. When determining the largest Snoman pad to use for a given location, the size of the Snoman pad is decreased until it meets the current spacing requirements or the Min Percent value is reached.
10. Select a Max Percent value. This controls the maximum size of the generated Snoman pad as a percentage of the host pad size.
11. Select an Offset value. This is the distance maintained between the center of the host pad and the edge of the Snoman pad being generated. This value can be negative.
12. When you are finished making your selections, click the OK button.
13. If you selected the Window option, you are prompted to create your selection window.
14. The data are processed, and any errors are logged to your report file, as well as to the Analysis area of the Navigator.

Tools|Teardrops

Toolbar Button: 

The Teardrop function creates a maximum material condition at the point of trace entry into a pad. All pad locations that a Teardrop cannot be generated for, are highlighted and their locations are noted in a report file.

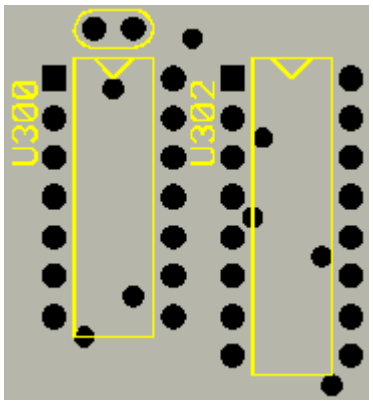
To add teardrops

1. A netlist is required for the Teardrop function to work properly. If one does not exist, use the Tools|Netlist|Generate command to create a netlist.
2. Select the Tools|Teardrops command. The Teardrops dialog box appears.
3. In the Report File box, specify the filename of a report file to log any errors. By default, the file is saved in the Samples folder. Click the Browse button to search for a different folder.
4. Select the Layer to process. If 0 is entered, all visible layers are processed.
5. Only flashes that use the specified D-codes will be processed. To select specific D-codes, click the Arrow button and select the D-Codes from the Select Apertures dialog box. If All is specified, all flashes are processed.
6. In the Percent Of Host box, specify the size of the Teardrop's tail on pads. The length of the teardrop's tail is calculated as a percentage of the host pad diameter. This value may be less than or greater than 100%.
7. In the Length Multiple box, specify the length of the Teardrop's tail on T junctions. This value is calculated as a percentage of the host trace width, and may be fractional.
8. In the Width Multiple box, specify the width of the Teardrop's tail on T junctions. This value is calculated as a percentage of the host trace width, and may be fractional.
9. A Teardrop will not be added if it violates either of the Minimum Spacing values. Pad/Trace is the minimum spacing that must be maintained between pads and adjacent traces, and Trace/Trace is the minimum spacing that must be maintained between adjacent traces.
10. Select Window if you only want to process data within a windowed area. Otherwise, the entire layer is processed.
11. If you previously ran the Teardrop function, and wish to delete any existing teardrops before creating new ones, select Delete Existing Teardrops.
12. When you are finished making your selections, click OK.
13. If you selected the Window option, you are prompted to create your selection window.
14. Teardrops are added to your specifications, and any errors are logged to your report file, as well as to the Analysis area of the Navigator.

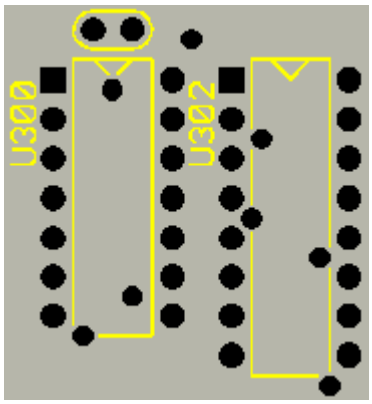
Tools|Fix SilkScreen

Toolbar Button: 

The Fix SilkScreen command automatically clears silkscreen data away from pads.



Before



After Running Fix SilkScreen

To run the Fix SilkScreen function

1. Make sure your layers are properly tagged in the Layer Table.
2. Select the Tools|Fix SilkScreen command. The Fixup SilkScreen dialog box appears.
3. In the Pad Layer box, select the layer that contains the pads you wish to clear the silkscreen data from. Alternatively, you can select an NC layer, and the function will clear the silkscreen from any drill data.

4. In the SilkScreen Layer box, select the corresponding silkscreen layer.
5. In the Min Spacing box, specify the minimum clearance between the pads and the silkscreen.
6. If you want to process only data within a selection window, select the Window option. If you want to process the extents of the layers, do not select this option.
7. Click the OK button. If you did not select the window option, the layers are processed.

   -or-

   If you selected the Window option, create a selection window around the desired data. The selected data are processed.

Related topics

Analysis|DRC/MRC

Tools|Pad Removal

This command allows you to remove isolated and stacked (redundant) pads.

Tools|Pad Removal|Isolated

Toolbar Button: 

Selecting this command will remove any unused pads from your inner layers.

To remove isolated pads

1. Make sure your layers are properly tagged in the Layer Table. Only layers tagged as Inner will be processed.
2. If you wish to process more than one inner layer, turn on the layers you wish to process, and turn off the inner layers you do not wish to process.
3. Select one of the Tools|Convert|Drawn Pads commands to convert all drawn pads to flashes.
4. Select the Tools|Pad Removal|Isolated command. The Isolate Pad Removal dialog box appears.
5. In the Layer box, select the layer to remove isolated pads from. Select 0 to process all visible inner layers.
6. In the D-code box, select the D-code that is used for the pads you wish to process. Select 0 to process all pads.
7. If you wish to only process a select window of pads, select the Window option. Otherwise, the entire layer is processed.
8. Click OK. If you selected the Window option, define your selection window. The pads are processed, according to your selections.

Tools|Pad Removal|Stacked

Selecting this command will remove any unnecessary pads that are stacked one on top of another, or are completely buried and on the same layer.

To remove stacked pads

1. Select one of the Tools|Convert|Drawn Pads commands to convert drawn pads to flashes.
2. If you wish to process more than one layer, turn on the layers you wish to process, and turn off the layers you do not wish to process.
3. Select the Tools|Pad Removal|Stacked command. The Stacked Pad Removal dialog box appears.
4. In the Layer box, select the layer to remove stacked pads from.
5. Define a Tolerance if you want any pads that are offset from each other within a certain amount to be considered stacked.
6. If you wish to only process a select window of pads, select the Window option. Otherwise, the entire layer is processed.
7. Click OK. If you selected the Window option, define your selection window. The pads are processed, according to your selections.

Tools|NC

The NC commands control the creation and optimization of NC data. To control the display of NC data in the workspace, as well as specify the dialect of the machine or controller you intend to use, select the Options|Configure command, and use the options under the NC Settings tab.

Tools|NC|Create Path From

Use this command to convert existing drawn data to mill paths. The existing data are not actually altered or deleted; the resulting mill path is placed on an NC layer.

If you wish to manually draw a new mill path, use the Add|Mill Path command.

To convert drawn data to a mill path

1. Use the Setup|Layers command to create an NC layer to place the mill path on, if you don't already have one.
2. Use the Setup|NC Tools command to define your mill tools.
3. Select the Tools|NC|Create Path From command. The Create Mill Path From dialog box appears.
4. Select a Destination NC Layer to place the mill data.
5. In the Tool box, specify the number of the tool that you wish to use to create the mill data.
6. Select Automatic Mode if you want to select a line segment, and have the system locate all connected lines to form the entire mill path. When using Automatic mode, set the Max Outline Item Gap value to specify the maximum amount of space that can exist between items and still be considered part of the same outline.

   -or-

   Select Interactive Mode if you want to manually select multiple individual line segments that compose the mill path.

   -or-

   Select Manual Mode if you want to select only one line segment to convert to a mill path.
7. If you are starting with segmented arcs and want to create "true" mill arcs, utilizing circular interpolation, select the Convert Segmented Arcs To True Arcs command (recommended). Otherwise, any arcs will continue to be represented by linear mill segments (when the path is routed, it will go much slower because the router will pause at the beginning of each segment).
8. If you select the Convert Segmented Arcs To True Arcs option, click the Arc Conversion Setup button to control how the arcs are created:

   Minimum Segments
   The minimum number of segments which must be within a valid segmented arc. The higher this number, the more accurate the system will be when selecting the arcs to process.

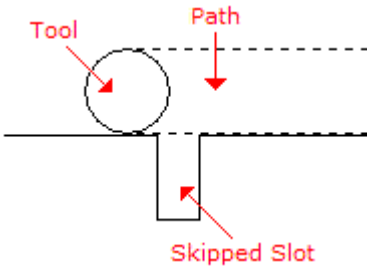
   Maximum Length
   The maximum length a line segment can be for it to be considered part of an arc. This prevents inclusion of traces as pieces of arcs.

   Short Seg Length
   "Short segments" can be present at the beginning and end of segmented arcs. This option specifies the maximum size of any short segments that might be found, which will be automatically appended to an arc. This is necessary because short segments can cause the arc-finding algorithm to fail to find valid arcs due to insufficient precision.

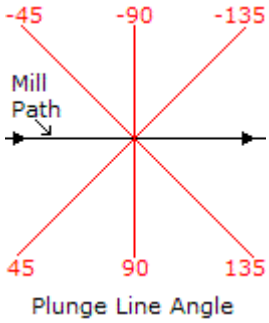
   Length Tolerance
   The tolerance for variation of line segment lengths for segments inside a segmented arc, not including the first and last segments (short segments).

   Angle Tolerance
   The maximum angle between a perpendicular line from the line segment, and the line from the midpoint of the given line segment, to the center of the arc, for the purpose of determining whether the segment belongs to the arc.

   Click OK to return to the Create Mill Path From dialog box.
9. A compensated mill path is offset from the centerline by the radius of the mill tool, or the value set in the Compensation Index Table in the NC Tool Table. If you wish to use compensation, select the Specify Compensation Direction option. You will be prompted later to select the offset direction.
10. If your board has any "slots" or "cut-outs" that are smaller than the diameter of the mill tool, the result would be a rout that cuts out more than the defined width of the slot. If you select the Skip Slots Less Than Tool Size option, the router will skip the small slots, as illustrated below.



11. Select the Double Path option if you want the mill tool to rout the same path twice. This can be used to clean any rough edges left behind from the first rout, especially when routing flexible circuits.
12. If you want to define a plunge point that lies outside of the mill path, specify an Angle and Length for the plunge line. The plunge point will be based on these settings and the vertex you designate later as the starting point of the path. The Angle is the deviation (counter-clockwise for positive angles, clockwise for negative angles) of the plunge line from the first segment of the mill path, or the tangent at the starting point (if the first segment is an arc). The angle is limited to a range of +/- 135 degrees. Specify a value of "0" for both options, if you want the plunge point to be on the mill path (at the starting point).



13. An overshoot is similar to a plunge line, in that it is used to create a smoother edge on the board, except that it applies to the tool extraction point. The last segment of the mill path is extended, so that the tool extracts at a location outside the board. If you want an overshoot, specify the Overshoot Length. Enter a value of "0" if you do not want an overshoot.

⚠ *Tip: If the start and end points are on a corner, and compensation is used, a gap may appear between the plunge and extraction points. The Overshoot and Plunge options are provided so that you can close this gap, but they must be used in an appropriate manner. The angle of the plunge is intended for situations where the start point is in the middle of an edge that you do not want to plunge into directly. A properly defined overshoot would close any gap, in this case. If the start point is on a corner, the plunge angle should be 0 so that compensation does not open a gap that the overshoot cannot close.*

14. Click the OK button.
15. If you selected Automatic or Manual Mode, click on any location in the drawn data that you wish to convert to mill. The line segments are highlighted.

    -or-

    If you selected Interactive Mode, select each of the line segments that you wish to convert to mill. When you have completed the path, press the End key.
16. If you selected the Specify Compensation Direction option, you are prompted to select the offset direction. Click on the side of the mill path where you want the offset applied.
17. Click on the starting point of the mill path.

    Your mill path is created. Note the following symbols are used to represent the plunge and extraction points.



Tools|NC|Drill Separate

This command allows you to move a set of drill hits from an NC layer to a new NC layer. The drill hits are selected by tool number and plating status.

To separate drills onto a new NC layer

1. Select the Tools|NC|Drill|Separate command. The Drill Separate dialog box appears.

2. In the Source Layer box, specify the NC layer that contains the drill hits you wish to move.
3. In the Destination Layer box, specify the empty layer that you wish to move the drill hits to.
4. Each tool is listed by number, with its size provided for your reference. The Plated and Non-plated columns contain check boxes for each tool. Select whether to move the plated or non-plated (or both) hits for each desired tool, by clicking on the appropriate check box next to each tool. (If no selection is made for a tool, none of the drill hits created by that tool will be moved.) To select all plated hits for every tool, click on the Plated column heading. To select all non-plated hits for every tool, click on the Non-plated column heading.
5. Click the OK button to move the drill hits.

Tools|NC|Drill Mask

This command creates the mask used for tenting your non-plated holes prior to plating.

To create a drill mask

1. Make sure your tools are properly defined as Plated or NonPlated in the NC Tool Table. For your NonPlated drills, specify a Mask Size.
2. Select the Tools|NC|Drill Mask command. The Create Drill Mask dialog box appears.
3. Specify your NC layer in the Source Layer box.
4. Specify an empty layer for your Destination Layer.
5. Click the OK button, and your drill mask is created.

Warning: If you change any drill data on the NC layer that you created the drill mask for, you will need to delete and recreate the drill mask.

Tools|NC|Set Group Order

Toolbar Button: 

All NC machines should to be told what order to create each drill hit or mill path. It is best to optimize your NC tooling paths so that they are as short and efficient as possible. If you want to group your drill/mill data so that sets of data are created in a specific sequence, you must first define the group order with this command. All NC data with a given group number are processed together, and the group numbers are processed from lowest to highest. Therefore, it is legal to assign the same group number to multiple items. Then use the Tools|NC|Optimize command to optimize the tooling path within each group. If you do not optimize the tooling path, the tooling within each group would happen in a completely random fashion, potentially causing your tooling to take more time to complete.

To set the NC group order

1. Make sure that NC group order numbers are visible in the workspace. Select the Tools|NC|Display Settings command. In the Configuration dialog box, select (check) the Display NC Group Order Number option.
2. It is recommended that you turn off the visibility of all your non-NC layers.
3. Select the Tools|NC|Set Group Order command. The Set NC Group Order dialog box appears.
4. Select the NC Layer that you would like to work on.
5. By default, all NC items are assigned the same sequence number, 999, so they will be processed together. This function allows you to specify items that you wish to be processed before or after the default setting. There are several ways to specify the initial group number you would like to use:

   Follow Initial Selection
   You select an item that you want all others to follow. After selecting OK in the Order Drill Items dialog box you will be prompted for the hit that subsequently selected hits will follow. For example, if you select hit 999 as your initial selection, the next hit you select will be 1000.

   Make Initial Selection First
   The items you select will start with the next lowest unused number in the sequence, and ascend from there. The number you are starting with is in parentheses.

   Make Initial Selection Last
   The items you select will start with the next highest unused number in the sequence, and ascend from there. The number you are starting with is in parentheses.

   Sequence Number
   You manually specify the group number of the next hit you select.
6. Select the Mark Items As Merge Data option to flag your NC data as "merged". Virtual Panelization will not step and repeat those merged items.
7. Click the OK button.

8. Use the Selection Filter to order the items as specified. The Status Bar displays "(N)ext, (P)rev, (F)irst, (L)ast, (G)oto", indicating that typing the first letter of these commands will display an item using the next, previous, first, last, or explicitly specified group number.
9. When you are finished, press the Esc key to exit the function.

Tools|NC|Optimize

Toolbar Button: 

All NC machines should be told what order to create each drill hit or mill path. If you do not optimize the tooling path, the tooling within each group would happen in a completely random fashion. There are two ways to optimize your data: You can create your tooling path based on the tool number (all data for Tool 1 is tooled first, then Tool 2, etc.), or you can group your data so that sets of data are created in a specific sequence.

To optimize your NC data

1. If you want to optimize your NC data based on NC group order numbers, first define your group order numbers with the Tools|NC|Set Group Order command.
2. It is recommended that you turn off the visibility of all your non-NC layers.
3. Select the Tools|NC|Optimize command. The Optimize NC Layer dialog box appears.
4. Select the NC layer that you wish to optimize.
5. You can have the system initially sort the NC data by Tool Number or NC Group Order Number. If sorted by Tool Number, the system repeatedly scans the board, looking at data created by one tool at a time (e.g. tooling is first sorted for Tool 1, then sorted for Tool 2, etc.) If sorted by NC Group Order Number, the system looks at data with the same group order number, optimizes the tooling, then looks at data with the next group number and optimizes it, etc.
6. Select how you would like the data optimized:

   X
   Specifies that the NC layer is to be optimized by performing an X sort. The quality of the X sort results are dependent on the sort data. Results are produced very quickly. If you select an X sort, you must also specify a Swath Width, which is the width of the system's scanning window when it looks at the data. The larger the swath width, the more NC data that will be scanned at one time, and more the tool can potentially deviate from a straight path in the X direction.

   Y
   Specifies that the NC layer is to be optimized by performing a Y sort. The quality of the Y sort results are dependent on the sort data. Results are produced very quickly. If you select a Y sort, you must also specify a Swath Width, which is the width of the system's scanning window when it looks at the data. The larger the swath width, the more NC data that will be scanned at one time, and more the tool can potentially deviate from a straight path in the Y direction.

   Wander
   The wander sort is performed using a nearest-neighbor method. Results are produced quickly.

   Advanced
   The advanced sort is performed using a simulated metal annealing method that produces excellent results when sufficient processing time is permitted. The time required to produce significant reductions in tooling time is very much dependent on the settings found in the Advanced NC Optimization dialog box and the CPU speed of your computer. For larger designs, using aggressive advanced sort parameters, optimizations can easily take hours to perform.

   The Advanced Sort Setup button opens the Advanced NC Optimization dialog box, where you can set the Advanced Sort parameters. In general, the correct values to use for advanced optimization depend both on how good the original path was, and how much time you wish to spend optimizing. If the original data are in fairly good order, you can use smaller values for the level duration and level change rate. If the NC locations in the original data are at scattered locations, larger values are recommended. If at any time during your optimization, you wish to stop the program, you can press the cancel button to stop the optimization and save the best result so far.

   You can use the scale to the optimization level. The optimization values are dynamically updated below. You can also select the Manual Override option to change the settings.

   Starting Level determines how large of a change the optimization will make to the original tooling in one step. Values larger than 100 will sometimes do a better overall optimization of the original tooling path, but will cause the optimization to take a lot longer to finish. Smaller values will cause the optimization to finish faster, but will usually not do as good of an overall job of optimization. Generally, this value should be left at 100.

Change Rate controls the number of steps the optimization will take, starting from the Starting Level. Larger values will improve the results of the optimization, but the results will take longer to produce. Likewise, smaller values will take less time but produce longer tooling paths. Extremely low values may even cause the resulting NC file to be longer than before optimization. Values of between 0.60 and 0.95 are recommended to produce well optimized files. Values greater than 0.999 are not allowed. The default value is 0.75.

Level Duration controls how much time the optimization will take at each of the steps entered in the change rate above. Larger values will cause the optimization to try more possible paths at each step, while smaller values will cause fewer paths to be tried. Values of between 100 and 1000 are recommended for this variable. The default value is 100.

X:Y Movement Speed Compensation is used to take into account the difference between the speed of the NC machine in the X or Y axis. The default values are 1 for each, meaning the tool travels the same speed in each direction. If for example, the tool on a particular machine travels twice as fast in the X direction than in the Y, you can set the X value to 0.5 to take this into account when optimizing your files.

7. After clicking OK in the Optimize NC Layer dialog box, the data are optimized according to your selections.

Tools|NC|Double Mill Path

Select the Double Mill Path command if you want the mill tool to rout the same path twice. This can be used to clean any rough edges left behind from the first rout, especially when routing flexible circuits.

To double-rout a mill path

1. Select the Tools|NC|Double Mill Path command. You are prompted to select the mill path to double-rout.
2. Using the Selection Filter to aid your selection, click on the desired mill path. Double-routing is immediately applied to it, with double-arrow graphics appearing on the path (where a normal path would have a single arrow graphic).
3. Click on any additional mill paths that you want to double-rout, or press the Esc key to exit the function.

Tools|NC|Display Settings

Toolbar Button: 

This command serves as a shortcut to the Options|Configure/NC Settings dialog box.

Tools|Test Points

The Test Points commands are used to create test fixture information for your design.

Bed of Nails Test Procedure

1. Setup test point parameters
2. Generate test points
3. Add additional test points (if necessary)
4. Stagger test points (if necessary)
5. Snap probes
6. Save test file

Flying Probe Test Procedure

1. Setup test point parameters
2. Generate test points
3. Add additional test points (if necessary)
4. Save IPC-D-356A file

See the Tutorials for complete information on test fixturing.

Tools|Test Points|Setup

Use this command to set up the type of testing you are performing, and the related layer and fixture probe parameters. By default, the layers labeled "Top" and "Bottom" are the layers tested. Before you begin, make sure your layers are properly tagged.

To set up your test information

1. Select the Tools|Test Points|Setup command. The Test Setup dialog box appears.
2. Select the Type of testing you are performing.
3. Select whether you are testing the top, bottom, or both sides, and enter the target layer(s) to place test point graphics on. These should be empty layers, a different one for each side.

4. Select the Use Top Solder Mask... or Use Bottom Solder Mask... option if you want to use the solder mask layers to determine which pads will ultimately be covered on the top or bottom layer; those pads will not be selected as test points.
5. If you selected Flying Probe test fixturing, you are finished (click the OK button). If you selected Bed Of Nails test fixturing, you need to select the parameters for your fixture plates and probes. Select the spacing between grid points (this is a single-density grid).
6. Click the Probe Setup button. The Probe Setup Table contains a list of probes assigned to pads that are created using the D-codes given in the list. By default, the probe with code 1 is assigned to all pads. To change the assignment, click on the desired Probe Code and type the new number in the text box.
7. To change the Probe Code settings, click on the Probe Properties button. The Probe Properties dialog box appears. The probe diameter, length and maximum deflection angle (the largest angle a probe can employ while testing a point) appear below the table. To change any values, click on any item in the list and type the new value.
8. When you are finished defining your probes, click OK.

The fixture parameters will be used when generating the fixture plate drill files for this design. If a tool of the exact size required is not found, the closest larger size tool will be used.

Tools|Test Points|Generate

Use this command to generate the test point locations. All points considered for testing, depending on what you specify in the Test Point Generation dialog box, will be added to the layer(s) specified in the Tools|Test Points|Setup command.

To create test point locations

1. If you have not already done so, use the Tools|Netlist|Generate to create a netlist.
2. Select the Tools|Test Points|Generate command. The Test Point Generation dialog box appears.
3. Select All to test all pads.

   -or-

   Select Ends Only to test only pads on the ends of nets. If you selected to output test points for both the top and bottom layers in the Test Point Setup, then the Double Sided option is available. If you select Double Sided, an additional check is made when determining if a given pad is a true end point; if a pad appears to be an end point, the system checks to see if it is a through-via (i.e. a through-hole drill pierces it or, if no NC drill layer is specified, the system checks if the aperture type is though-hole and a through-hole aperture exists at the same location on the opposite layer). If the pad is found to be a through-via, it is considered a "mid point" and not an "end point", and a test point will not be placed on it. Only the true ends on either side of the board are tested.
4. Select Ignore Isolated if you wish to overlook isolated pads.
5. Click OK when you have completed your selections. Your test points are generated.

If you wish to delete any or all of the test points, either select the Edit|Undo command immediately after generating test points, or use the Edit|Delete command and set the Selection Filter to delete only flashes on the layer where the test points are placed.

Tools|Test Points|Add

This allows easy adding of additional test points, after you have run the Tools|Test Points|Generate command.

To add individual test points

1. Select the Tools|Test Points|Add command. The Add Test Points dialog box appears.
2. If you want to select individual test point locations by clicking on pads (i.e. flashes), select the By Flash option. If you want to add test point locations to all pads/flashes on a selected net, select the By Net option
3. If you have a double-sided test fixture, you must select whether you want to add test points to the top or bottom layer. The target layer number and name is displayed in the Add Test Points dialog box, for your reference.
4. If you selected By Flash, click on the pad that you wish to place a test point on. The test probe that is placed is based on the parameters in the Test Point Setup. However, if you selected to use a solder mask for test point accessibility, note that an accessibility check is not performed. Also, if a test point already exists at that location, a new one is not added.

   -or-

   If you selected By Net, click anywhere on the desired net, and test points are added to all flashes of the net, on the specified side, that do not already have test points. If you selected to use a solder mask for test point accessibility in the Test Point Setup, then an accessibility check is performed before test points are added.

To delete any test points, either select the Edit|Undo command immediately after adding test points, or use the Edit|Delete command and set the Selection Filter to delete only flashes on the layer where the test points are placed.

Tools|Test Points|Stagger

Automatic test point staggering is used to provide additional spacing between test probes. As an example, fine pitch SMT components typically require test point staggering to insure there is no probe-to-probe contact. It does this by offsetting the location of the test point from the center of the corresponding pad.

To stagger test points

1. After generating your test points, select the Tools|Test Points|Stagger command. The Stagger Test Points dialog box appears.
2. If you created a double-sided test fixture, select the Side To Stagger. This is the layer that contains the Test Point locations to be probed. The target layer number and name is displayed for your reference.
3. Specify the Probe Spacing, which is the minimum spacing allowed between probe points.
4. Specify the Edge Spacing. This is the minimum distance allowed between a staggered probe location and the edge of a pad. When a probe point is moved by the stagger function, it will not move a probe closer than this distance to the edge of a pad.
5. Specify a Max Flash Size. The function will not attempt to stagger probes used on pads that are larger than the specified size. If 0.0 is specified, all test points are eligible for staggering.
6. If you select the Use Middle option, the staggering function will place points in the middle of pads in addition to the edges, as long as doing so does not violate the probe spacing value. If you do not select this option, all points will be staggered to the edges of the pads.
7. If you wish to stagger only test points within a selection window, select the Window option. Otherwise, the entire layer will be processed.
8. The Initial Direction options allow you to select the direction of the staggering. For example, if the setting was Left/Up, the first point that needs to be staggered is moved either to the Left or Up depending on the location of the pad which caused this point to be staggered. The next point that is to be staggered is then moved in the opposite direction, and so on.
9. Click the OK button. If you did not select the Window option, the test points are staggered, and you are finished.
10. If you selected the Window option, define your selection window in the workspace. The test points within the window are staggered.
11. You may define additional selection windows, or press the Esc key to exit the function.

Tools|Test Points|Analysis

The Analysis command checks existing test points and reports any errors in the Analysis section of the Navigator. If you selected the Double Sided option when you used the Tools|Test Points|Generate command to create the test points, then both the top and bottom layers are processed at the same time. If the Double Sided option was not selected, the top and bottom layers are processed independently.

The Analysis process checks for at least two test points for each net; if there are less than two test points for a given net, one of two types of errors will be generated:

- If there are at least two flashes (pads) on the net, then a "Missing Test Point" error is generated. This error can be fixed by the system, by right-clicking on the error in the Navigator and selecting the Fix command. This will add a test point to each pad on the net that does not already have a test point. If solder mask accessibility was specified in the Test Point Setup, the test point will only be added if the flash is accessible.
- If there are less than two pads in the net, then a "Too Few Flashes" error is generated. This error cannot be fixed by the system.

When you highlight a test point error in the workspace (using the right-click menu in the Navigator), the associated net is highlighted. An "X" appears at each test point location, and a circle appears around any pad that does not have an associated test point.

Tools|Test Points|Flying Probe

The Flying Probe commands are used for exporting your test files for flying probe test machines.

Tools|Test Points|Flying Probe|IPC-D-356A

The IPC-D-356A format is an intelligent netlist file that is commonly used for flying probe testing.

To export an IPC-D-356A netlist

1. Select the Tools|Test Points|Flying Probe|IPC-D-356A command. The Export IPC-D-356A dialog box appears.

2. In the File Name box, enter a name for your netlist file. By default, the file is saved in VisualCAM's Samples folder. Use the Browse button to choose a different folder or file name.
3. Enter the number of the NC layer that contains your through-hole drill information (this is the same NC layer that was used when the netlist was generated). If no NC layer is available, the system will look at the aperture table "type" field to determine whether a pad is SMT or not.
4. Select Include Unconnected Pads Using N/C Net Name to have all unconnected (isolated) pads output using the IPC-defined net name "N/C". Otherwise, all unconnected pads will be ignored.
5. Select Include Solder Mask Information if you want to include data indicating solder mask usage. If no solder mask is present, then this option is not available. (If you have a solder mask but this option is not available, your solder mask layer is not properly tagged in the layer table.)
6. The Export Conductor Data option exports the trace data for the top and bottom layers.
7. IPC-D-356A does not support raster fill polygons. If you want polygon information maintained with your trace data, select the Vector Fill Polygons And Export With Conductor Data option to make the polygons vector fill. Also specify the size of the lines to use for the solid vector fill.
8. Export Net To Net Adjacency Data to report any nets within your specified distance to each other. (In the netlist, each net is reported along with any other net that is adjacent to it, according to the distance you specify.) Select this option, if desired.
9. Select the Export Probe Points option, and whether you want to export points for the top or bottom side layers. Normally the netlist would only include the center points of the net locations. In cases where the probes are offset from the center of the pad, such as when the probes have been staggered, the Export Probe Points option reports the exact probe test location.
10. Click the OK button to export your file.

Tools|Test Points|Test Fixture

The Test Fixture commands are specific to Bed of Nails testing.

Tools|Test Points|Test Fixture|Snap To Grid

After you have set up your test fixture information and added your test points, it is time to "snap" the test points to the fixture plate grid. This tells VisualCAM where to create the drill holes on the fixture plates.

When you select this command, the probes are snapped to the grid. A message box appears, telling you if the system was unable to map any of the probes.

Tools|Test Points|Test Fixture|Save Files

This command allows you to specify all the parameters defining the probes and fixture plate requirements. Up to seven fixture plate drill files can be created. A netlist file and grid-to-board mapping report file are also generated.

To generate bare board test fixture files

1. Select the Tools|Test Points|Test Fixture|Save files command. The Test Fixture File Creation dialog box appears.
2. The Grid plate is the plate closest to the fixture machine, and the Board plate is the plate closest to the PCB. The Guide plates are aids for guiding the probes to the correct location on the board. Select the plates you wish to output. If you are testing a double-sided board, these definitions are for the top side.
3. Enter the desired fixture drill file names for the plates in the Filename box for each. By default, the files are saved in the VisualCAM Samples directory. Click the browse buttons to save the files in a different location.
4. In the Dist boxes, enter the distance of the each selected plate to the board.
5. In the Thick boxes, enter the thickness of the fixture plates.
6. By default, the file is saved in the native VisualCAM .nl format. To save the netlist as an IPC-D-356A file, select the IPC-D-356A option.
7. Enter the desired Netlist File name. By default, it is saved in VisualCAM's Samples folder. Click the Browse button to save to a different folder.
8. The Grid/Board Map file is a simple ASCII report file that shows the probe assignment for each location, the coordinates of the test point, and corresponding probe grid location. Select a name for this .txt file.
9. Click the NC Format button to view or change the format used for the test point drill file. The NC Export Format dialog box appears.

   Dialect
   The format of the file to export. Available options are Excellon, Excellon 2, Sieb & Meyer 1000, Sieb & Meyer 3000/5000, and Takeuchi

M.N
This is the coordinate format. For example, a 2.3 format specifies two decimal digits before an implied decimal point and three following. (12250 represents 12.250 if the coordinate format is 2.3).

Mode
Absolute mode is when all X:Y coordinates are referenced to a common origin. Incremental mode is when each X:Y coordinate is a displacement from the previous coordinate.

Zero Suppression
No zero suppression allows coordinates to have zeroes in front or back of integers. Leading zero suppression shows zeroes only at the end of a number, trailing zero suppression shows zeroes only at the beginning of a number.

Special
You may enable Modal mode which will reduce the size of your files by removing all redundant draft codes and coordinates. Metric outputs the file in metric format. The default is imperial (inches).

Character Set
Indicate the character text used in the file.

Set As Default
Saves the current settings as the default settings for this command.

10. Click the Tool Setup button if you wish to change any of the tool information in the NC Tools table. See the Setup|NC Tools command topic for details on modifying the table.
11. If you are testing a double-sided board, click the Bottom Side button, and repeat steps 2 through 10 to define the files for the bottom side (this button changes to "Top Side" at this point, allowing you to switch between both sides at will).
12. When you are finished, click OK to create the files.

Tools|Embedded Passives

The Embedded Passives commands are used to reverse engineer drawn and flashed data into intelligent embedded passive devices. This allows you to then check the data for errors, create reports, etc.

Tools|Embedded Passives|Find

The purpose of this function is to tell the system what conductive layer is paired with what embedded passive layer, and have it automatically "build" the embedded passives from the information on those layers. If you have multiple embedded passive layers, you must repeat this function for all the embedded passive-conductive layer pairs in your database.

Each of the three main options in this command (converting the drawn resistor shapes to flashes, marking them as resistor shapes, and building the embedded passive) can also be performed manually using the following commands, respectively:

Tools|Convert|Drawn Pads

Tools|Embedded Passives|Mark Resistor Flashes

Tools|Embedded Passives|Build Embedded Passive

To automatically find embedded passives

1. Use the Setup|Layers command to organize and tag your layers.
2. Select one of the Tools|Convert|Drawn Pads commands and convert all drawn pads on your conductive layers to flashes. If this is not done, the system will not be able to properly build your embedded passives.
3. Select the Tools|Embedded Passives|Find command. The Find Embedded Passives dialog box appears.
4. Select the Embedded Passive Layer and Conductive Layer pair you wish to work with.

Warning: If you have already associated a conductive layer with the selected embedded passive layer, you will be warned that the conductive layer is being changed and all previously defined embedded passives will be removed.

5. In the Material Constant box, specify the Ohms Per Square value of the material used in your resistors.
6. Select the Layer option to have all the data on the selected layers processed, or select the Window option to only process data within a selection window.
7. Select the Convert Drawn Resistor Shapes To Flashes option. This analyzes the drawn data on your passive layer, determines which draws form a rectangular shape (rectangles or squares), and converts them to flashes. Any drawn data that does not appear to form a rectangular shape is not converted. To increase the system's ability to compensate for variances in drawn data, you can specify a Tolerance value. By doing so, you relax the criteria for determining whether a shape is rectangular; this compensates for things like slightly inconsistent edges due to the

aperture shape used to draw the rectangle. The larger the tolerance, the more the drawn shape can stray from that of a true rectangle and still be converted. Usually a value of 0.0005 inches is sufficient.

8. Select the Identify Resistor Flashes option to tag each rectangular and custom aperture flash on the embedded passive layer as a resistor shape. This is automatically done if you select the Build Embedded Passive Resistor Objects option.
9. If you have previously run this function, and wish to remove all previously identified resistor shapes, select the Clear Previously Identified Flashes option.
10. Select the Build Embedded Passive Resistor Objects option. This matches the resistor shapes on the passive layer to their termination bars on the conductive layer. The resistor shapes must touch, or be within the specified Tolerance of the termination bars, in order for them to be matched.
11. Click the OK button. If you selected the Layer option, all your data are processed. If you selected the Window option, define the selection window in the workspace, and the data are processed. You can select additional data, or press the Esc key to end the function.

    Every flash that is tagged as a resistor on the passive layer is now marked with an Ohms symbol.



Resistor Flash

Every embedded passive that has been built is highlighted with an outline.



"Built" Embedded Passive

A list of all your embedded passives appears in the Valid Embedded Passives list in the Navigator, by reference designator. You can view information about each embedded passive, and perform a variety of tasks. See the Embedded Passives topic for details.

If you have flashes that are marked as resistors, but do not belong to a valid embedded passive, the termination bars fell outside your defined tolerance. They are listed in the Invalid Embedded Passives list in the Navigator, and you will need to manually construct the embedded passive using either the Construct Embedded Passive command in the Navigator's right-click menu, or the Tools|Embedded Passives|Build Embedded Passive command.

Prior to exporting your embedded passive data, it is recommended that you use the Analysis|DRC/MRC command and run the embedded passive analyses. This will ensure that your data are properly aligned, sized, etc.

Tools|Embedded Passives|Mark Resistor Flashes

The Tools|Embedded Passives|Find command allows you to have the system automatically mark all the flashes on your passive layers as resistor shapes. You can also manually mark each flash as resistor shapes, using the Mark Resistor Flashes command.

⚠ *Tip: This can also be done in the Navigator. See the Embedded Passives topic.*

To manually mark resistor shapes

1. Use the Setup|Layers command to organize and tag your layers.
2. Select the Tools|Embedded Passives|Find command. In the Find Embedded Passives dialog box, select only the Convert Drawn Resistor Shapes To Flashes option. (Do not select Identify Resistor Flashes or Build Embedded Passive Resistor Objects.)

   -or-

   Select one of the Tools|Convert|Drawn Pads commands to convert the drawn resistor shapes on your passive layers to flashes

   After converting your data to flashes, note that all the flashes now appear Navigator, under Embedded Passives/<passive layer name>/Non-Embedded Passive Flashes.
3. Select the Tools|Embedded Passives|Mark Resistor Flashes command.
4. Click on each flash that you want marked as a resistor shape. Every flash that is tagged as a resistor on the passive layer is marked with an Ohms symbol. Press the Esc key when you are finished.



Resistor Flash

After marking your flashes as resistor shapes, note that they now appear in the Navigator under Embedded Passives/<passive layer name>/Invalid Embedded Passives. Use the Tools|Embedded Passives|Build Embedded Passive command to finish defining your embedded passives.

Tools|Embedded Passives|Build Embedded Passive

The Tools|Embedded Passives|Find command allows you to have the system automatically identify the embedded passives in your design. You can also manually identify each embedded passive, using the Build Embedded Passive command.

Tip: This can also be done in the Navigator. See the Embedded Passives topic.

To manually identify embedded passives

1. Use the Setup|Layers command to organize and tag your layers.
2. Select the Tools|Embedded Passives|Find command. In the Find Embedded Passives dialog box, select the Convert Drawn Resistor Shapes To Flashes and Identify Resistor Flashes options (Do not select Build Embedded Passive Resistor Objects.)

   -or-

   Select one of the Tools|Convert|Drawn Pads commands and convert the drawn resistor shapes on your passive layers to flashes. Then select the Tools|Embedded Passives|Mark Resistor Flashes command, and mark each resistor shape.

   After marking your flashes as resistor shapes, note that they now appear in the Navigator under Embedded Passives/<passive layer name>/Invalid Embedded Passives.
3. Select the Tools|Embedded Passives|Build Embedded Passive command. You are prompted to select a resistor shape.
4. Click on a resistor shape. You are prompted to click on its associated termination bars.
5. Click on each of the termination bars. You are prompted to enter an embedded passive reference designator. This is a unique alphanumeric name used to identify each embedded passive in the Navigator.

   Every embedded passive that has been built is highlighted with an outline, and appears in the Navigator under Embedded Passives/<passive layer name>/Embedded Passives.



"Built" Embedded Passive

Tools|Embedded Passives|Etch Compensation

In order to achieve the correct finished size for etched passives, the manufacturer has to factor in tolerances for the etching process. This function oversizes etched passives by a specified amount, on the sides that are perpendicular to the termination bars.

To apply etch compensation to passives

1. This function only works on layers tagged as Passive Sub. Use the Setup|Layers command to make sure your layers are properly tagged.
2. Use the Tools|Embedded Passives|Find command or Tools|Embedded Passives|Build Embedded Passive command to build your embedded passives.
3. It is recommended (but not required) that you run the embedded passive analyses in the Analysis|DRC/MRC command, to ensure that your data are properly sized, aligned, etc.
4. Select the Tools|Embedded Passives|Etch Compensation command. You are prompted to select the flash for a resistor shape.
5. Using the Selection Filter, select one or more resistors to apply compensation to.
6. In the dialog box that appears, enter the amount of compensation to apply, and click OK.
7. Select additional flashes, or press the Esc key to exit the function.

Tools|Embedded Passives|Query

See the Query|Embedded Passive command for a description of this dialog box.

Tools|Embedded Passives|Report

See the Documentation|Reports|Embedded Passives command for a description of this dialog box.

Tools|Layer Spread

Toolbar Button: 

Use this command to reduce your film costs by automatically copying and spreading selected layers onto one layer, and thus one sheet of film.

To create a layer spread

1. Use the Options|Configure/Display command to make sure the film box is the required size.
2. If you want your layers in any particular order, use the Layer Table to organize them.
3. Select the Tools|Layer Spread command. The Layer Spread dialog box appears.
4. Select Auto Mode to have the system automatically calculate the positioning of the images in the film box. Otherwise, you will be in Manual Mode and must specify the number of rows and columns in which to place the images.
5. In Auto Mode, the Edge To Edge Spacing options specify the edge-to-edge spacing between the resulting images. In Manual Mode, the Point To Point Spacing options specify the spacing from a point on one image to the same point on the next image.
6. In the To Layer box, select a layer to place the layer spread. An empty layer is recommended.
7. If you are in Manual Mode, specify the number of Cols. (columns) and Rows that you want in your layer spread.
8. Select the Row Major option if you want your images ordered across each row. In a 4x3 layer spread, the layers would be spread as illustrated below.

Layer 9	Layer 10	Layer 11	Layer 12
Layer 5	Layer 6	Layer 7	Layer 8
Layer 1	Layer 2	Layer 3	Layer 4

-or-

Select the Col Major option if you want your images ordered up each column. In a 4x3 layer spread, the layers would be spread as illustrated below

Layer 3	Layer 6	Layer 9	Layer 12
Layer 2	Layer 5	Layer 8	Layer 11
Layer 1	Layer 4	Layer 7	Layer 10

9. Click the OK button. The Layer Spread dialog box appears.
10. Click on each layer that you want in the spread, and click OK.
11. The layout of your layer spread is displayed in the workspace, and you are asked to confirm that the layout is correct. If you click Yes, the layer spread is created. If you click No, the function is aborted.

Tools|Stencils

These commands are used on a paste mask layer, to enhance the openings over the pads for improved solder paste application. Once you have defined your desired stencil shapes using the Tools|Stencils|Setup Shapes command, and assigned those shapes to openings using the Tools|Stencils|Manual Conversion command, you have subsequently defined an automated process for future stencil conversion. The shapes and assignments are automatically stored, and can be accessed in the future using the Tools|Stencils|Automatic Conversion command.

Tools|Stencils|Setup Shapes

This function is used to enhance the finished shapes for paste stencil openings. You are taking an existing paste stencil, and modifying the shapes to those that meet the *IPC-7525 Stencil Design Guidelines*.

To define stencil shapes

1. Select the Tools|Stencils|Setup Shapes command. The Stencil Shapes Setup dialog box appears.
2. To add a new stencil shape definition, click the New button. The New Stencil Shape dialog box appears.
3. Enter a descriptive name for the shape in the Shape Name box.

4. In the Shape Type list, select the type of shape you want to define. A preview of the selected shape appears in the small display to the right.
5. The Justification option controls the placement of the new shape in relation to the pad you assign it to. If Justification is not selected, the shape will be centered on the pad. If Justification is selected, you need to select which edge of the shape will be justified against (i.e. lined up with) the corresponding edge of the pad.

Tip: Justification is only available if the top and/or bottom conductive layers associated with your paste mask layers are currently loaded and tagged.

6. Normally, when the shapes are assigned to paste openings, the protrusions/insets point to each other according to the orientation of the pad pair. The Reverse Direction option will make the protrusions/insets point away from each other.
7. If you select to set sizes by Percentage, you are defining the enhanced shapes that original paste openings will be converted to based upon the size of the original opening, and what the size of any protrusion, inset, or corner radius will be relative to the finished size of the enhanced opening. You are not defining the overall size of the finished paste opening. The existing opening that you later associate the enhanced shape to will determine the size of the finished opening. If you select to set sizes by Amount, then you are defining an exact reduction or enlargement of the original opening, using the current units of measure.

   If you want the shape to be smaller than the original opening it is associated with, the numbers used for the Increase/Decrease Height and Width should be negative (preceded by a "-"). If you want the shape to be larger than the associated original opening, the number should be positive.
8. The elements of the shape you have to define depend upon the Shape Type you select and whether you are setting sizes by Amount or Percentage. If a corner radius applies, you can select to include the specified radius amount to the radius of inset corners, or leave those corners right angles.
9. Click OK to save your stencil shape.
10. Repeat steps 2 through 8 for any additional shapes you want to define. You can also use the Edit and Delete buttons to modify or remove shape definitions, respectively.
11. When you are finished, click the OK button to close the dialog box. The shape definitions are stored in a global table that will be available for future use.

After you have defined your stencil shapes, you can use the Tools|Stencils|Manual Conversion command to assign those shapes to paste openings associated with pad pairs (and subsequently convert those openings). You can also save those assignments to a file for future use by the Tools|Stencils|Automatic Conversion command.

Related topics

Creating Stencils

Tools|Stencils|Manual Conversion

After you have assigned your desired stencil shapes to different paste openings associated with pad pairs, the system will remember the stencil shapes and assignments for future jobs. Those future jobs can be run using the Tools|Stencils|Automatic Conversion command.

To assign shapes to pads and convert them

1. Use the Tools|Stencils|Setup Shapes command to define your desired opening shapes.
2. Make the paste mask layer, that contains the shapes you wish to convert, the active layer. To make the process easier, it is also recommended that you turn off the visibility of any other unnecessary layers.
3. Select the Tools|Stencils|Manual Conversion command. You are prompted to select your first pair of openings to convert.
4. Select the upper-left and lower-right corners of a selection window around the desired pair of openings. The Stencil Shapes Setup dialog box appears.
5. Select the desired shape, and click OK. The system will create the association of that shape to the selected openings, and scan the layer for other pairs of openings that match. The match is determined by the size of the openings and pitch (center-to-center spacing).
6. A message box appears, telling you how many matching pairs were found. If you want to convert all of them now, click Yes. The openings are modified. If you click No, the openings will not be converted, but the opening-shape association you created is saved. You can convert the openings later using the Tools|Stencils|Automatic Conversion command.

Tip: If you want to maintain the existing data on your paste mask layer, and put the enhanced shapes on an empty layer, click No when you are prompted to convert the openings. Use the Automatic Conversion command instead, where you can specify a separate destination layer for the data.

7. You are prompted to select another pair of openings. Repeat steps 4-6, as desired. Otherwise, press the Esc key to exit the function.

Tools|Stencils|Automatic Conversion

Automatic Conversion can be used if you have already defined stencil shapes using the Tools|Stencils|Setup Shapes command, and assigned those shapes to specific pairs of stencil openings using the Tools|Stencils|Manual Conversion command.

To convert pad shapes

1. Make sure your paste mask layers are properly tagged in the Layer Table.

Tip: If there is any stencil data that you what the conversion process to ignore, create a Select Group of those items.

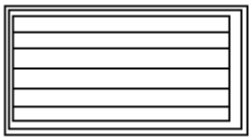
2. Select the Tools|Stencils|Automatic Conversion command. The Automatic Stencil Conversion dialog box appears.
3. If you have already assigned stencil shapes to openings, those assignments are listed. You can Save those assignments for use in future designs by clicking the Save button. If you want to load assignments from a previously-saved .ini file, click the Load button.
4. Select the desired shape conversions from the list by checking the box next to the Stencil Shape.
5. The system automatically selects the available paste mask layers as the Source/Destination layers. You can add or delete layers from the list by right-clicking on the desired line and selecting the appropriate command from the menu.
6. By default, the resulting enhanced shapes are placed on the same layer as the source data. If you wish to place the enhanced shapes on a different layer, click on the desired Destination Layer and select a different layer from the list. You can also change the Source Layer by clicking on it, if desired.
7. If you have a Select Group of openings that you do not want processed, select the Don't Convert Items In Select Group option.
8. When you are finished making your selections, click the Run button to run the conversion.

Tools|Convert

The Convert commands are used to optimize data, or simply convert one type of item to another.

Tools|Convert|Drawn Pads

Many CAD systems still output rectangular pads, such as surface mount pads, using multiple draws to fill in the rectangle rather than a more efficient single flash. This results in bloated Gerber files and increased processing times. Also, it is virtually impossible for high-level CAM functions, such as DRC, to recognize these drawn entities as a pad rather than just a bunch of traces. The example below shows the difference between a typical drawn pad and a comparable flash.





The drawn pad shown requires 27 separate Gerber commands to accomplish what one Gerber flash can accomplish. As you can see, if you have 2000 of these drawn flashes, you will have a Gerber file with at least 54,000 lines when flashes could accomplish the same thing in only 2,000!

It is recommended that the conversion of drawn pads to flashes be the first thing done to your designs. This will usually ensure complete and trouble free conversion. Also, you must convert all drawn pads to flashes *before* generating a netlist or running most other CAM functions.

Tip: If you are reverse engineering embedded passives, and need to convert drawn resistor shapes to flashes, we recommend you use the Tools|Embedded Passives|Find command.

If you have mask layers with your data, you can automatically (and more quickly) identify drawn pads for conversion with the Tools|Convert|Drawn Pads|Automatic command.

If you do not have any mask layers, use the Tools|Convert|Drawn Pads|Select Draws to convert your drawn pads to flashes.

Tools|Convert|Drawn Pads|Select Draws

Toolbar Button: 

This function converts drawn pads to flashes, based upon data that you select for conversion. If you have mask layers with your data, you can automatically (and more quickly) identify drawn pads for conversion with the Tools|Convert|Drawn Pads|Automatic command.

To convert draws to flashes by manually selecting draws

1. Select the Tools|Convert|Drawn Pads|Select Draws command.
2. Make sure, among your other Selection Filter settings, that the Type is set only to Draw.
3. Select one occurrence of the drawn pad that you wish to convert. The Convert Drawn Pads dialog box appears.

   The system locates all drawn pads identical to the one you selected. The Calculate Pad Size area displays information on the selected pads to be converted.
4. Select how you want the draws replaced.

   Create True Size/Shape Custom Apertures
   Select this option if you wish to have selected items converted to a single custom aperture with the same appearance.

   Create Standard Apertures
   Select this option if you wish to have selected items converted to one aperture of the specified shape and size. Select the Shape and Size of the aperture you wish to use. If an aperture of that size and shape already exists, it is used. Otherwise, a new aperture is created. If you also select the Replace Patterns Rotated 90 Degrees option (see step 5), then up to 3 apertures may be created.

   Specify D-code
   Select this option if you wish to have the selected items converted to an existing aperture. Specify the new D-code number for the pad created by the conversion. To view the Aperture Table, where you can create or edit apertures as necessary, click the Edit Apertures button.
5. Select the Replace Patterns Rotated 90 Degrees option if you wish to have items of the same shape, which are rotated 90 degrees, converted also.
6. Select the Use Net Connectivity option if you wish to only convert pads that are on the same net as the example pad that you selected.
7. Select the Include Isolated Items option if you wish to have items that are not part of a net (isolated pads) checked for possible conversion.
8. To increase the system's ability to recognize pads that are the same size, and account for minor variances in drawn data, you can specify a Tolerance value. By doing so, you relax the criteria for determining which pads are the same size; pads whose sizes are within the tolerance value are determined to be the same. Usually a value of 0.0005 inches is sufficient.
9. After you make your selections and click OK, all occurrences of any matching drawn pads are located and you are asked to confirm whether to continue.

Tools|Convert|Drawn Pads|Automatic

Toolbar Button: 

The automatic drawn pad conversion requires a past or solder mask layer with a corresponding top or bottom layer. For example, a "Mask Top" or "Paste Top" layer requires a "Top" layer to exist. The mask layer must have data on it, but it is not required that the mask data be flashed. The openings on the mask layer are used to look for drawn pads to be converted to flashes.

If you do not have any mask layers, use the Tools|Convert|Drawn Pads|Select Draws command to convert your drawn pads to flashes.

To automatically convert draws to flashes

1. Select the Tools|Convert|Drawn Pads|Automatic command. The Automatic Drawn Pad Conversion dialog box appears.
2. The Control Layer is the layer number of the solder or paste mask layer that corresponds with the top or bottom layer that you wish to convert pads on. Click the down-arrow button to select the desired layer (to prevent an incorrect layer from being specified, you cannot type a number directly into the box).

3. If the selected Control Layer is a paste mask layer, the Paste Mask Expansion option allows you to enter a value by which paste mask openings will be oversized. Oversizing the openings may be required, because often paste mask openings are smaller than the corresponding pads on the top/bottom layer.
4. Click the Select Layers button to specify one or more layers that contain the drawn pads you wish to convert.
5. Specify the Maximum Size, in the X and Y dimensions, that a group of draws may be, and still be considered for conversion to a flash. If a group of draws exceeds the maximum size, they will not be converted.
6. The Tolerance controls how close the replacement aperture must be in size to the original draws, to be considered a valid replacement. If no replacement is found, then an error will be logged.
7. Select the Replace Patterns Rotated 90 Degrees option if you wish to have items of the same shape, which are rotated 90 degrees, converted also.
8. Select the Window Control Layer Data option if you want to convert only pads in a specific area. This is useful for designs that have text blocks or other data outside the actual board area, as excluding that extraneous data will speed up the process.
9. Click the OK button. If you did not select the Window Control Layer Data option, the conversion process begins immediately. If the option was selected, you must select two points to define the area to be processed.
10. When the process is complete, a message is displayed. If you selected the Window Control Layer Data option, you may select another area to process, or press the Esc key to end the function.

    If no appropriate replacement aperture is found to replace a drawn pad, then an error is logged. Automatic drawn pad conversion will only find and convert round, oblong, rectangle, and square shapes. "No Aperture" errors are generated for shapes that are not recognized; you will need to use Tools|Convert|Drawn Pads|Select Draws to convert them. An error is also logged if a mask opening exists, but no drawn data was found in the corresponding location on the conversion layer.

Tools|Convert|Drawn Text

Toolbar Button: 

This function allows you to convert drawn text to "true text". Although this function may be used to convert any drawn text on any layer, its primary purpose is to convert drawn reference designators on your silkscreen layer, so that you can use your silkscreen to reverse engineer parts for assembly purposes. As such, the function has certain characteristics that enhance the conversion of reference designators, which may not apply to the conversion of other text.

For more information on the entire reverse engineering process, see the *Reverse Engineering for Assembly* tutorial.

To convert drawn data to text

1. Select the Tools|Convert|Drawn Text command. the Drawn Text Conversion dialog box appears. This dialog box contains 3 areas, accessible by tabs: "Conversion Control", "Alphabetic Characters", and "Digits and Symbols".
2. Under Conversion Control, select the layer number of your Source Layer. By default, your silkscreen layer is selected. If you wish to convert the text on a different layer, click the arrow button to select it from a list.
3. The Destination Layer can be an empty layer (the default), or it can be the same as your Source Layer. If you make it your Source Layer, the drawn text on your silkscreen will be replaced with the resulting text.
4. Select the Convert Rotated Text option if you want the system to look for text that is rotated.
5. Select the Text Is Mirrored option if the Source Layer is also a backside layer (the text appears mirrored in the workspace).
6. The Tolerance controls how close the replacement text must be in size to the original draws, to be considered a valid replacement. If no replacement is found, the draws are not converted.
7. The Character Set Definition features allow you to load, save, and search for files containing definitions of which characters are associated with which set of draws. This is useful if the text you are converting is identical from job-to-job.

    If you have a Character Set Definition file from a previous job that you wish to use, click the Load button and select the file. If you want the system to search a folder containing multiple Character Set Definition files, and determine which one is most appropriate for converting the text on the Source Layer, click the Search button and identify the folder that the files are in.

    -or-

    To identify each character in your silkscreen and tell the system what it is, continue with the steps in the section below, *Defining a Character Set*.

8. Once you have defined a character set, you need to tell the system which Font to use for the resulting text. By default, it will use the GerbTool-Stroke Font of the same size as the largest character defined in the character set. If you wish to use a different font, or change the size of the GerbTool-Stroke font, click the Select Font button and select your desired font and the applicable dimensions of the font.
9. If you want to convert only draws that are in a specific area of the Source Layer, click the Window Area To Convert option. This is useful if you have different character definition sets that you want to use to convert text on the Source Layer. Otherwise, the entire layer is searched.
10. Click the Convert button. If you selected the Window Area To Convert option, you are prompted to define the windowed area. The system searches the Source Layer for text to convert.
11. Instead of just converting one character at a time, the system groups together your text into strings. Since the primary purpose of converting drawn text is for converting reference designators, this function is designed to insure that reference designators that are placed close together are not accidentally grouped into the same string. Strings consisting of only alphabetic characters ("ABC"), numeric characters ("123"), alphabetic characters followed by numeric characters ("ABC123") or numeric characters followed by alphabetic characters ("123ABC") are automatically converted to text. When strings consisting of multiple groups of alphabetic and numeric characters ("AB12CD" or "12AB34") are found, the system zooms to it in the workspace, and the String Validation And Correction dialog box appears.

    You can split the reference designators apart by manually placing them on separate lines in the dialog box, or (since reference designators typically are letters followed by numbers) click the Split Numeric | Alpha button to have the system separate the text between a number followed by a character. For example, if "C70R62" is identified, clicking the button will separate the text into "C70" and "R62". You can have the system separate the text like this automatically by selecting the Automatically Split Mixed Alphanumeric Strings... option.
12. When the conversion process is complete, a confirmation message appears. The new, true text now appears in the workspace. You can now reverse engineer your data, for assembly, using your silkscreen.

Defining a Character Set

If you select Manual Identification, you are required to create a window around each character, and tell the system what it is. If you select Automatic Identification *(the recommended process)*, you specify parameters that the system will use to automatically find characters for you to identify.

When a character set is used to convert drawn text, you specify a single font and size to use for the true text. Therefore, when defining a character set, you should only include characters of approximately the same size. For example, if you ultimately only want to convert reference designators to text, do not identify characters in title blocks or other non-reference designator text at the same time. Otherwise, the size differences will be lost during the conversion, and you will have some true text characters that are larger than the original drawn text. If you want to convert text of different sizes, and maintain those differences, you must define separate character sets for them.

Similarly, if there are drawn characters using different fonts, and you wish to maintain the different font in the conversion to true text, define separate character sets for the different fonts.

Automatically

1. Click the Define button in the Drawn Text Conversion dialog box. The Character Set Definition dialog box appears.
2. Select the Automatic Identification option.
3. The Source Layer can be the same silkscreen layer that you specified in the main Conversion Control settings, or a different layer that you want to use for creating the character definitions (at this point, you are only creating the character definitions, not actually converting the draws).
4. Specify the Height and Width of the largest character you wish to identify. This helps the system filter out draws that are not characters.

    If you are unsure of the height and width, use the Query|Measure|Point To Point command to measure a largest character in the workspace that you wish to identify in this character set.
5. Select the Text Is Mirrored option if the Source Layer is also a backside layer (the text appears mirrored in the workspace).
6. Click the OK button. The system analyzes the Source Layer, and if characters were found, the Character Confirmation And Identification dialog box appears. You now have to teach the system what character each set of draws represents.

    If no characters were found, verify that you have specified an appropriate Height and Width for your characters. If the values are too small, nothing will be found.
7. The system highlights an identified character in the workspace, and shows a copy in the Drawn box. The copy should appear in the correct (upright, non-mirrored) orientation in the Drawn box.

If the character does not appear correct, select the Mirror or Rotate option (and specify the angle of rotation) as appropriate. When the character appears correct, type the text character it corresponds to in the Character box (using the correct capitalization), and click OK. The system asks you to identify another character.

If the system has identified something that is not actually a character, just click the Ignore button to skip it and move on to another character to identify.

Do not be concerned if the system asks you to identify several characters that are the same. You may know that all those draws are the same number or letter, but the system does not! It means that each of the characters that it asked you to identify have slight variations in the way that they were drawn. It is in your best interest to identify each of these, as it means that the ultimate character conversion will be that much more accurate.

8. When the system has gone through all the sets of draws that it found, a confirmation message appears that tells you how many character variations were identified. You are returned to the Drawn Text Conversion dialog box.

   If you want to see what characters have been defined in the selected character set, click the Alphabetic Characters or Digits And Symbols tab at the top of the dialog box. You can see every text character and the corresponding drawn characters that have been associated with them (they are split into two tabs merely because of space limitations in the dialog box). When multiple definitions for the same drawn character exist, then arrow buttons are displayed beside the drawn character so that you can see all of them. The drawn characters are all displayed at the same scale, based on the largest character defined in the character set, so drawn characters will appear to be different sizes in the display.

   The characters in the character set may be modified or deleted by clicking on the drawn character. A shortcut menu appears, with two options. If you select Change, the Character Confirmation And Identification dialog box appears (See Step 5 of the Manual procedure, below). If you select Delete, the selected drawn character is deleted from the character definition set. (Use caution, as deleting a character cannot be undone with the Edit|Undo command.)

   If any characters were missed, they can be added by using the manual method described below, or rerunning the automatic method with different parameters.

9. If you want to save the character set definition for future use in other jobs, or you wish to define multiple character sets for this job, click the Save button and save it as a .csd file in the VisualCAM CharDefs folder.
10. If you need to define another character definition set for drawn characters of a different size or font, repeat steps 1-8. Otherwise, you may now complete the conversion process. Continue with Step 8, in the above procedure.

Manually

1. Click the Define button in the Drawn Text Conversion dialog box. The Character Set Definition dialog box appears.
2. Select the Manual Identification Option, and click the OK button.
3. You are now prompted to create a window around a drawn character in the workspace. Use the Selection Filter as an aid, if necessary. By default, you are placed in Windwo+Xing mode. Click on a corner of a selection window, and then on another corner to surround a drawn character. The character will be highlighted as soon as it is selected.

Tip: Use the arrows, +, and - hotkeys to pan and zoom to a desired character. You want to make sure that all the draws associated with a character are selected, or you may end up with undesirable results when you perform the conversion to text.

   If not all of the desired drawn data is selected, you can continue to select data until everything you want is highlighted. Note that the selecting data works as a toggle when selecting drawn characters. If any data is already highlighted (selected), and you select it again, then highlighting is removed and it is no longer selected. This is useful if more draws are selected than you intended (for example, part of an adjacent character is also highlighted). You can also press the Esc key to gradually un-select data.

4. When you have selected the drawn character, click the End button. The Character Confirmation And Identification dialog box appears. You now have to teach the system what character that set of draws represents.
5. A copy of the drawn character appears in the Drawn box. Make sure the character appears in an upright, non-mirrored orientation. If the character is mirrored or rotated, select the Mirror or Rotate option (and specify the angle of rotation) as appropriate. When the character appears correct, type the text character it corresponds to in the Character box (using the correct capitalization), and click OK.
6. Repeat steps 3-5 to identify all the drawn text.
7. When you are done, click the Esc key and return to the Drawn Text Conversion dialog box.

   If you want to see what characters have been defined in the selected character set, click the Alphabetic Characters or Digits And Symbols tab at the top of the dialog box. You can see every text character and the corresponding drawn characters that have been associated with them (they are split into two tabs merely because of space limitations in the dialog box). When multiple definitions for the same drawn character exist, then arrow buttons are displayed beside the drawn character so that you can see all of them. The drawn characters are all displayed at the

same scale, based on the largest character defined in the character set, so drawn characters will appear to be different sizes in the display.

The characters in the character set may be modified or deleted by right-clicking on the drawn character. A shortcut menu appears, with two options. If you select Change, the Character Confirmation And Identification dialog box appears (see Step 5). If you select Delete, the selected drawn character is deleted from the character definition set. (Use caution, as deleting a character cannot be undone with the Edit|Undo command.)

If any characters were missed, they can be added by using the automatic method described above, or rerunning the manual method.

8. If any characters were missed, they can be added by using the manual method described below, or rerunning the automatic method with different parameters. If you want to save the character set definition for future use in other jobs, or you wish to define multiple character sets for this job, click the Save button and save it as a .csd file in the VisualCAM CharDefs folder.
9. If you need to define another character definition set for drawn characters of a different size or font, repeat steps 1-8. Otherwise, you may now complete the conversion process. Continue with Step 8, in the above procedure.

Tools|Convert|Arcs To Segmented Arcs

Toolbar Button: 

This command will convert circular interpolated (true) arcs into line segments. Use this command if your photoplotter does not support circular interpolation (true arcs).

When operating on Mill data, the difference between this and the Segment Arcs option during File|Export|NC (Drill/Mill) is this function allows you to control which arcs are segmented and the at which chord angle.

To convert arcs to segments

1. Select the Tools|Convert|Arcs To Segmented Arcs command. The Segment Arcs dialog box appears.
2. The Chord Angle is the angle at which the line segments will deviate from each other. The smaller the angle, the more line segment are drawn, and the smoother the curve will be. Select your desired chord angle, and click OK.
3. Using the Selection Filter, select one or more arcs to convert.

   When selecting an arc in a mill path for conversion, if the selection mode is "Window", then the entire mill path must be within the selection window, and all arcs within the mill path are converted. To select specific arcs within a mill path, either "Item" or "Window +Xing" mode must be used.

   If you select a milled circle, because milled circles are predefined operations within a mill program, it is converted to a mill path.
4. You are asked to confirm your selection. When you click Yes, the arcs are converted.
5. Select additional arcs to convert, or press the Esc key to exit the function.

Tools|Convert|Segmented Arcs To Arcs

Toolbar Button: 

This command will convert arcs made up of line segments to circular interpolated (true) arcs. Converting segments to arcs will help reduce the size of your database files.

To convert line segments to arcs

1. This command can process entire layers. If you have several layers with segmented arcs that you wish to convert, make those layers visible, and turn off any layers you do not want to process.
2. Select the Tools|Convert|Segmented Arcs To Arcs command. The Segmented To Real Arcs dialog box appears.
3. Specify the name and location of the report file to be generated by the command. By default, it is saved in VisualCAM's Samples folder. Click the Browse button to save to a different folder.
4. In the Layer box, select the layer that contains the arcs you wish to convert. "All" or "0" processes all visible layers.
5. Specify the minimum number of segments which must be within a valid segmented arc. The higher this number, the more accurate the system will be when selecting the arcs to process.
6. Specify the Maximum Length a line segment can be for it to be considered part of an arc. This prevents inclusion of traces as pieces of arcs.
7. "Short segments" can be present at the beginning and end of segmented arcs. The Short Seg. Length option specifies the maximum size of any short segments that might be found. This is necessary because short segments can cause the arc-finding algorithm to fail to find valid arcs due to insufficient precision.

8. In the Length Tolerance box, specify the tolerance for variation of line segment lengths for segments inside a segmented arc, not including the first and last segments (short segments).
9. In The Angle Tolerance box, specify the maximum angle between a perpendicular line from the line segment, and the line from the midpoint of the given line segment, to the center of the arc, for the purpose of determining whether the segment belongs to the arc.
10. If you select the Attach Short Segments To Lines option, short segments will be attached to each other and longer segments for the purpose of eliminating these from the database.
11. If you wish to return the dialog box settings to their system defaults, click the Defaults button.
12. Click the OK button when you are finished. All the arcs on the specified layers that qualify for conversion are automatically converted into arcs.

Tip: To check if an arc has been converted, use the Query|Item command. A true arc will be identified as such in the Item Properties display.

Tools|Convert|Gerber To NC

Toolbar Button: 

This command converts all drawn and flashed graphics on a layer to NC data. If any polygons are converted, their outline becomes a mill path.

Tip: For increased control over the properties of mill paths, we recommend you use the Tools|NC|Create Path From command.

If you wish to copy individual graphics to an NC layer, use the Edit|Copy command.

Converting Gerber to NC

1. If you have any drawn pads that you wish to convert to drill hits, use one of the Tools|Convert|Drawn Pads commands to convert the pads to flashes.
2. Use the Tools|Pad Removal|Stacked command to remove any redundant pads.
3. Select the Tools|Convert|Gerber To NC command. The Gerber To NC dialog box appears.
4. Specify the Source Layer of drawn and flashed data which are to be converted to NC data.
5. Specify the Destination Layer to place the new NC data. This can either be an existing NC layer, or an empty layer.
6. If you select an empty Destination Layer, select the NC Tool Table to assign to the Destination Layer. If you select an existing NC layer, the tool table that is already assigned to that layer is displayed, and cannot be changed.
7. Click the OK button. Your data are converted.

Tools|Convert|NC To Gerber

This command will convert drill and mill data from an entire NC layer to flashes, draws, and polylines (as appropriate) on a specified non-NC layer. If round apertures that match the size of the NC tools do not exist, new apertures are created.

If you wish to convert only selected drill hits, use the Edit|Copy command.

To convert NC data to Gerber

1. Select the Tools|Convert|NC To Gerber command. The NC To Gerber dialog box appears.
2. In the Source Layer box select the NC layer with the drill and/or mill data to be converted.
3. In the Destination layer box, select the layer that is to receive the converted data.
4. Click the OK button, and your data are converted.

Tools|Convert|To Custom

Toolbar Button: 

This command converts existing database items into a custom aperture. By nature, custom apertures are graphical entities. Therefore, once converted, they are assigned to their own D-code, and do not contain any references to the previous layer or D-code assignments of the items they were created from.

An exception to this is NC tool assignments. Since custom apertures are created automatically when importing drill files that contain Step and Repeat commands, tool assignments are internally stored in custom apertures. To change the tool assignments within an existing custom aperture, use the Custom Aperture Editor.

To convert data to a custom aperture

1. Turn on only the layers that contain the items you wish to convert to a custom aperture.

2. Select the Tools|Convert|To Custom command.
3. Using the Selection Filter, window around the data to convert to a custom aperture. The Gerber To Custom dialog box appears.
4. Specify a name for the custom aperture.
5. Select the Replace Original Selection option if you want to automatically replace your selected data with the resulting custom aperture. If you do not select this option, the custom aperture will still be created, but the original data will remain untouched.
6. Click OK. The items are immediately converted to a custom aperture, and assigned to a D-code.

Tip: You may expand (convert) one or more instances of a custom aperture back into its equivalent layer data using the Edit|D-code|Explode Customs command.

Tools|Convert|Raster To Vector

This command converts existing raster-filled polygons on the selected layers to vector-filled polygons.

To convert raster polygons to vector polygons

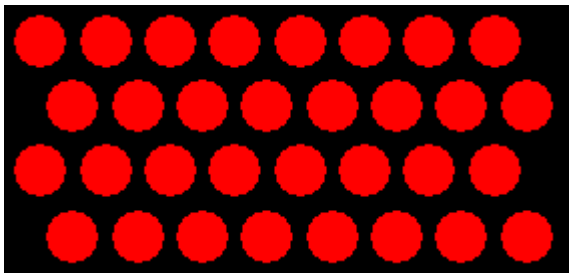
1. This function only processes visible layers. Turn on any layers you wish to process, and turn off those you don't.
2. Select the Tools|Convert|Raster To Vector command. The Raster To Vector Polygon Conversion dialog box appears.
3. Click the arrow button next tot he Layer box to select which layer you want to process. Select All Visible to process all visible layers.
4. The Outset selections determine how the outer boundary of the vector polygon relates to the original raster polygon. If you select Enable, the raster polygon's outer edge becomes the centerline of the vector polygon's border. If you select Disable, the vector polygon border's outer edge follows the raster polygon's outer edge as closely as possible (the vector data cannot completely get into interior corners because it is being drawn).
5. For the Border D-code, select the D-code to use to draw the border of the polygon.
6. There are four options for the Vector Fill Pattern:

   None creates a polygon outline with no fill.

   Solid creates a polygon with a solid fill. The Fill Dcode is the D-code that is used to create a solid polygon fill. Horizontal specifies that horizontal fill lines are generated, Inset specifies that the edge of the polygon is inset with the width of the Fill D-code until the entire area is filled.

   Hatched creates a hatched polygon fill pattern. Select (check) the number of lines to use in the pattern, then specify the D-code to be used for each line. Step Size values are used to specify the line to line spacing. Angle values are used to specify the angle of the lines used.

   Dotted creates a dotted polygon fill pattern. Select the D-code to use for the dots (flashes), and the spacing between the dots in each row (the X Spacing) and column (the Y Spacing). Specify whether the spacing should be measured from Center-to-Center or Edge-to-Edge. To create a staggered pattern of dots, select the Row Stagger option and decide which row should be offset. To have them staggered like the below example, the Row 2 offset should be half the value of the Y Spacing amount, and the Row 1 offset should be zero.



7. The Difference Detection option is a post-processing analysis check. If selected, it checks for differences between the raster polygon and the resulting vector polygon. Any differences that are greater than the specified minimum size are flagged as errors in the Analysis area of the Navigator.
8. Click OK to perform the conversion.

Tools|Convert|Draws To Polylines

This command converts all contiguous, drawn line segments on visible layers into single polyline entities.

To convert draws to polylines

1. Turn on the layers you wish to be processed, and turn off any layers that you do not want processed.

2. Select the Tools|Convert|Draws To Polylines command. All visible layers are immediately processed, and you are informed how many draws were converted, and the number of resulting polylines.

Tools|Convert|Composite To Layer

This function converts a composite of layers into a single, standard layer containing only positive data. You can also use this function to convert a single, mixed polarity layer into a layer with only positive data.

To convert a composite to a single layer

1. Select the Tools|Convert|Composite To Layer command. The Composite To Layer dialog box appears.
2. Click the arrow next to the Source Composite box to select the composite you wish to convert. The Select Layers dialog box appears. All the composites in your design are listed. If you wish to see the layers contained in each composite, click the + icon, or double-click on the composite name.
3. Click on the checkbox to select the composite you wish to convert to a layer, and click OK.
4. Click the arrow next to the Destination Layer box, to select the layer on which you want to place the data from the converted composite. The Select Layers dialog box appears, with a list of all the empty layers in your design.
5. Select a layer, and click OK.
6. Click OK in the Composite To Layer dialog box, and the composite layers are converted.

   None of the original composite information or layers is deleted. If you wish to delete the composite, select the Setup|Composite command. If you wish to delete the original layers as well, select the Setup|Layers command. You can also use the Navigator to perform the layer conversion and deletion.

Tools|Convert|Mill To Drill

This command converts milled objects to drilled objects. Any type of milled circle is converted to a drilled circle. Mill paths are converted to drill hits by changing straight segments to slots, and arc segments to a series of drill hits approximating those generated for a drilled circle.

Tip: This command is also available from the right-click shortcut menu, when you right-click on the appropriate data items.

Converting Mills to Drills

1. Select the Tools|Convert|Mill To Drill command. You are prompted to select the mill data to convert.
2. Using the Selection Filter, select one or more milled items to convert.
3. You are asked to confirm your selection. When you click Yes, the items are converted.
4. Select additional mill data to convert, or press the Esc key to exit the function.

Tools|Convert|Drill To Mill

This function converts drilled objects to milled objects. Drilled circles are converted to CCW Inside milled circles. Drilled slots are converted to mill paths.

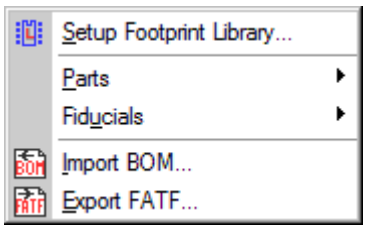
Tip: This command is also available from the right-click shortcut menu, when you right-click on the appropriate data items.

Converting Drills to Mills

1. Select the Tools|Convert|Drill to Mill command. You are prompted to select the drill data to convert.
2. Using the Selection Filter, select one or more drilled items to convert.
3. You are asked to confirm your selection. When you click Yes, the items are converted.
4. Select additional drill data to convert, or press the Esc key to exit the function.

Assembly Menu

The Assembly commands are used to reverse engineer part and fiducial information for your data, for assembly. These functions *cannot* be used to add intelligence to Gerber data for the purposes of exporting to a CAD system database.



Assembly|Setup Footprint Library

Toolbar Button: 

The Footprint Library contains information on all part footprints and devices defined for the current database. If you want to alter a single part instance (like changing its reference designator, device type, rotation, or to delete it), use the Navigator.

Tip: This function is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

Footprint List
All footprints are listed alpha-numerically, by footprint name. To view information about a footprint and its associated devices, click on the plus icon next to the footprint name. Most of the footprint and device information in the right column can be changed by clicking on it. Note that if you do change any of the information, it will affect all associated part instances that you have placed on your board.

Add Device
To add a new device to a footprint, click on a footprint name, and then click the Add Device button. You can choose to keep the default information that the system created for the device, or specify the name and the Reference Designator Prefix you want associated with the device that you just identified. The other information (Part Number, Value, Tolerance and Description) are optional. All parts have a color-overlay assigned to them, with the pins appropriately numbered, and the centroids remained marked with a target of the same color. A bounding box is placed through the centerline of the pins. You can select a color by clicking on the Display Color field (where it says "Click to Set"). You can now define a part instance on the board, for that device.

Delete Footprint/Device
The Delete button changes depending upon the item selected. If you want to delete a footprint, click on the footprint name in the list, then click the Delete Footprint button. Note that if you delete a footprint that is used, then all part instances using that footprint will be deleted as well (this will not affect your board data--it only affects part identification).

If you want to delete a device, click on the device name, and click the Delete Device button. If the device is used, you are given the opportunity to replace all part instances for that device with another device. If you click on a device that is the only one associated with a particular footprint, notice that the button says "Delete Footprint". That is because if you delete the only device associated with a footprint, you will delete the footprint as well.

Set Pins
To change the pin numbers for a part, click the Set Pins button. In the dialog box that appears, click on a pin, and a shortcut menu appears. You have the choice of designating the pin as Pin 1, or giving it a unique pin number (which can be alpha-numeric). Otherwise, you can click the Set Default Order button to assign a system-default order to the pin numbers, or click the Reverse Order button to reverse the current pin number order. The Swap Rows and Cols button is used on pin grid arrays that have alpha-numeric pin naming conventions based on rows and columns. Click the button if you need to swap the column-row naming sequence.

Load
Click this button to load a VisualCAM footprint library (.fpl) file.

Save
Click this button to save a footprint library (.fpl) file. If you have standard part definitions, then you can use this library file to reverse engineer other boards.

Assembly|Parts

The Parts commands are used for identifying, deleting, querying and creating reports for parts in the current database.

Assembly|Parts|Identify Using

These commands are used to reverse engineer part information for your data, for assembly. These functions *cannot* be used to add intelligence to Gerber data for the purposes of exporting to a CAD system database.

Assembly|Parts|Identify Using|Footprint Library

Toolbar Button: 

VisualCAM has the ability to save and load its own footprint library file. If you have a standard footprint library that you use for multiple boards, you can use it in reverse engineering.

Tip: This command is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

To Identify Parts Using a Footprint Library

1. All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, select Tools|Convert|Drawn Pads to convert them to flashes.
2. Select the Setup|Footprint Library command. The Footprint Library dialog box appears.
3. Click the Load button, and select the appropriate .fpl file to import.
4. Click the OK button in the Footprint Library dialog box.
5. Select the Assembly|Parts|Identify Using|Footprint Library command. The Part Identification dialog box appears.
6. Specify a Tolerance to account for small variances between part instances, and for misalignment in a row of pins during the pin identification process.
7. Select whether to find parts on the Top Layer, Bottom Layer, or both.
8. Select the Select Footprints option if you only want to look for parts with specific footprints in the library. Click OK. The Part Information dialog box appears, and individual part identification process begins. The process starts with the largest footprint in the library.
9. The pins for a detected part instance are highlighted, with each pin number displayed. Confirm that the Device, Reference Designator, and rotation are correct in the Part Information dialog box. If it is, click the Yes button. If not, either edit the information in the dialog box, or click the No button if the pins are not correct and you want the system to look again and pick a different set of pins. Click the Skip button if you simply want to skip the currently selected pin configuration for now (the most common reason being if you didn't run the Tools|Convert|Drawn Pads function in the beginning, and you notice that you have some drawn pads). Unlike pressing the No button, when you press Skip, the system will not continue to look for a "correct" pin configuration at that location.

   View Controls are provided in the Part Information dialog box. Since you cannot use the Hotkeys to zoom and pan around in the workspace, use the View Controls to get a better view of your parts.

   If you click the Cancel button, it will skip finding any more instances of the current part footprint on the board, and go to the next footprint in the library.
10. When no more part instances are found, a message appears, and you are finished.

Related topics

Reverse Engineering for Assembly

Assembly|Parts|Identify Using|IPC-D-356 Data

Toolbar Button: 

Reverse engineering from IPC-D-356 data is the most automatic and reliable method. However, note that your IPC-D-356 file must contain reference designators and pin numbers.

Tip: This command is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

To Identify Parts Using an IPC-D-356 File

1. All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, select Tools|Convert|Drawn Pads to convert them to flashes.
2. Select the File|Import|Netlist|IPC-D-356 command and import the netlist that goes with your data.
3. Use Edit|Align Layers and make sure your netlist layer is aligned to your layer data.
4. Select the Assembly|Parts|Identify Using|IPC-D-356 Data command. You are warned if you have not compared your netlist to your data, to check for any discrepancies. Although running a netlist comparison is recommended, it is not necessary to complete the reverse engineering process.

The system scans the database, constructs the Footprint Library, and identifies the part instances in the workspace. Any errors are logged in the Analysis area of the Navigator, and may ultimately be put in a .txt report file. When the process is finished, you are given the option of viewing the report file.

Tip: Pin connectors may be identified in the IPC-D-356 file for the purposes of the netlist, but they will show up as errors during reverse engineering because they are not components. These errors should be ignored.

Related topics

Reverse Engineering for Assembly

Assembly|Parts|Identify Using|SilkScreen Layer

Toolbar Button: 

You can use silkscreen layers to reverse engineer your part data for assembly. This function requires that the reference designators in your silkscreen be "real text", so you will have to do some processing of your silkscreen layer.

Tip: This command is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

To Identify Parts Using a Silkscreen Layer

1. All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, select Tools|Convert|Drawn Pads to convert them to flashes.
2. Select the Tools|Convert|Drawn Text command to convert your reference designators on your silkscreen layers to "real text".
3. Select the Assembly|Parts|Identify Using|SilkScreen Layer command. The Part Identification dialog box appears.
4. Specify a Tolerance to account for small variances between part instances, and for misalignment in a row of pins during the pin identification process.
5. Specify your Thru-Hole Drill Layer.
6. Select whether to find parts using the Top Silk Screen Layer, Bottom Silk Screen Layer, or both.
7. During the part identification process, you will be asked to identify the pins that correspond with each reference designator in the silkscreen. Select the Find Additional Part Instances With Same RefDes Prefix option if, after you identify each part footprint, you want the system to automatically identify any parts that the system finds with an identical footprint, that also share the same Reference Designator prefix.
8. Click OK. The system checks to see if any duplicate reference designators exist on the silkscreen, and if so, you have the option to generate and view a report with the reference designators and their X:Y locations.
9. You are now prompted to select the pins for the first reference designator. Using the Selection Filter and +/- keys to zoom in or out, if necessary, select the first pin in the part that corresponds to the highlighted reference designator.
10. Select the pin that is farthest from Pin 1 (typically the opposite diagonal corner of the part). If the part has more than 2 pins, the system automatically finds other pins in between. If any are missed or selected by mistake, click on the pins to select/deselect them. Use the Selection Filter, if necessary.
11. Press the End key when you are finished. The Footprint Name dialog box appears.
12. The system provides a default name for the footprint you just created, using the type of device, number of pins, and the iteration of the type of footprint that it is. SMD means that it is a Surface Mount Device, THD means it is a Thru-Hole Device. For example, THD6_2 is a Thru-Hole Device with 6 pins, and it is the second footprint of that type found. You can keep the default footprint name, or assign your own, and click OK.
13. The Device Name dialog box appears. You can choose to keep the default information that the system created for the device, or specify the name, number (optional), and the Reference Designator Prefix you want associated with the device that you just identified. All parts have a color-overlay assigned to them, with the pins appropriately numbered, and the centroids remained marked with a target of the same color. A bounding box is placed through the centerline of the pins. You can select a color by clicking on the color in the dialog box, or accept the system default.
14. If you selected Find Additional Part Instances With Same RefDes Prefix, if the system finds other pads that appear to use the same footprint as the part you just created, it highlights the pins and you are asked if they should be identified as another part. The system uses the unused reference designator that is located closest to the pins. If the suggested information is correct, click Yes to identify that part. If the part information is not correct, you can specify a new device name to associate with it, or rotate the part. If the identified reference designator is not correct for the highlighted pins, click No. The system will move on to any additional footprint matches that it detects, if any.
15. You now repeat the process to identify the remaining parts. When no more parts remain, the process ends.

    If you click Cancel in the Device Information dialog box, then you are prompted to select the pins associated with the highlighted reference designator again. If you want to abort the process entirely, press the Esc key. The

process aborts, but any part information remains. If you select the Assembly|Parts|Identify Using|SilkScreen Layer command later, you will have the option to Ignore the parts that have already been identified (basically starting where you left off), or to Delete the existing parts and starting over. (Note that if you delete the existing parts, the footprint information that you created will still remain in the library.)

Related topics

Reverse Engineering for Assembly

Assembly|Parts|Identify Using|Centroid File

Toolbar Button: 

Part Centroid files can be used to reverse engineer part data for assembly. A converter file is also necessary, which is used to instruct the system how to read the data in the various columns of the centroid file. If you do not have a converter file, you can easily create a new one using the interactive Centroid File Converter.

Tip: This command is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

To Identify Parts Using a Centroid File

1. All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, select Tools|Convert|Drawn Pads to convert them to flashes.
2. Select the Assembly|Parts|Identify Using|Centroid File command. The Use Centroid File dialog box appears.
3. In the Filename box, specify the centroid file to use. Click the browse button if you need to search for the desired file.
4. Select the correct Centroid File Converter for the centroid file from the Format list. These converter files are located in the CentroidConv folder in your main VisualCAM folder. If a converter does not exist, see Creating a Centroid File Converter for instructions.
5. Click OK. The Part Centroid Information dialog box appears.
6. Specify a Tolerance to account for small variances between part instances, and for misalignment in a row of pins during the pin identification process.
7. Specify the Thru-hole Drill Layer.
8. If you select Detect Additional Instances Of A Part, once a footprint is identified, all other parts that share the same device name or part number will be identified automatically and their associated reference designators assigned to them.
9. To aid in the visibility of the parts during the footprint identification process, select the Turn Layers On And Off Based On Part's Side option. This will make only the layer that the current part is on visible in the workspace.
10. As an additional aid, you can select the Keep Corresponding Silkscreen Layer On Top option to have the silkscreen data appear on top of the other data in the workspace (this does not affect the layer order).
11. The centroid of the first identified part is displayed as a target in the workspace. Click on Pin 1 for that part.
12. Select the pin that is farthest from Pin 1 (typically the opposite diagonal corner of the part). If the part has more than 2 pins, the system automatically finds other pins in between. If any are missed or selected by mistake, click on the pins to select/deselect them. Use the Selection Filter, if necessary.
13. Press the End key when you are finished. The Footprint Name dialog box appears.
14. The system provides a default name for the footprint you just created, using the type of device, number of pins, and the iteration of the type of footprint that it is. SMD means that it is a Surface Mount Device, THD means it is a Thru-Hole Device. For example, THD6_2 is a Thru-Hole Device with 6 pins, and it is the second footprint of that type found. You can keep the default footprint name, or assign your own, and click OK.

You will find that as you add footprints to the library, the Footprint Name dialog box does not appear after you identify the pins. This is because the system has matched an existing footprint to the part.

15. If you selected Detect Additional Instances Of A Part, you are told if any other part instances were found. Click OK, and the system moves to the next part to identify.

    All parts have a color-overlay assigned to them, with the pins appropriately numbered, and the centroids remained marked with a target of the same color. A bounding box is placed through the centerline of the pins.
16. You now repeat the process to identify the remaining parts. When no more parts remain, the Part Centroid Information dialog box disappears.

    You can skip a part at any time by clicking the Skip button in the Part Centroid Information dialog box.

    If you click Cancel, the entire process aborts, but any part information remains. If you select the Assembly|Parts|Identify Using|Centroid File command later, you will have the option to Ignore the parts in the

centroid file that have already been identified (basically starting where you left off), or to Delete the existing parts and starting over. (Note that if you delete the existing parts, the footprint information that you created will still remain in the library.)

Creating a Centroid File Converter

VisualCAM provides an interactive Centroid File Converter Creator that allows you to quickly create a centroid file converter file.

1. In the Use Centroid File dialog box, click the New Converter button. The Create Centroid File Converter dialog box appears. The name of the centroid file being imported appears at the top, and your current centroid file is displayed in the Format 1 tab.
2. Your centroid file appears in a spreadsheet, and default column names are provided at the top. These columns are not automatically matched to the centroid file; you must reorder and resize the columns to match the centroid file contents.
   - If your centroid file has columns in a fixed-width, with no consistent delimiter (instead of tabs or commas, a combination of single, and/or multiple spaces are used--or no spaces at all in some columns--or any number of other text characters are used), select the Use Fixed Width Columns option.
   - To move a column header, click it with your mouse and drag it to the correct location.
   - The column header width should exactly match the width of the centroid file columns. Click on the left or right edge of the column header and drag it to the appropriate location.
   - If you wish to delete a column header, right-click on it and select Delete Column from the shortcut menu. Note that once you delete a column, it can always be added again later (it appears as an Add command in the right-click menu).
   - If you have a column of information in your centroid file that you wish to be ignored, right-click on the column header currently above it, and select Add Skip Column from the shortcut menu. A header titled "Skip" now appears above the contents.
3. If your centroid file uses standard column delimiters (the Fixed Width option does not apply), specify the appropriate Delimiter.
4. Select the Units of measure for your centroid file.
5. If you have any lines in the centroid file, such as headers comments, specify in the Initial Lines To Ignore box how many lines at the beginning of the file should be ignored by the converter. Also specify, in the Comment Text box, the symbol that leads any comment lines that may be intermixed with your centroid information.
6. As you identify the columns in your centroid file, VisualCAM automatically lists recognized centroids in the Matched Centroids list. All recognized centroids are also colored red in the Centroid Format view.
7. When you are finished, click the Save Converter button to save your ACR file in the CentroidConv folder. The dialog box closes and your converter appears in the Format list of the Use Centroid File dialog box.

Related topics

Reverse Engineering for Assembly

Assembly|Parts|Identify Using|Select Footprint

Toolbar Button: 

If you do not have an IPC-D-356 netlist, centroid file, footprint library, or silkscreen layers for your database, you can manually identify parts using this function. For the sake of speed and accuracy, if you do have any one of the aforementioned data files, it is recommended that you use it to reverse engineer your part data for assembly.

This function can also be used after you have used any of the other Part Identification commands, and had to correct any data errors that required deleting a part instance and recreating it.

Tip: This command is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

To Manually Identify Parts

1. All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, select Tools|Convert|Drawn Pads to convert them to flashes.
2. Select the Assembly|Parts|Identify Using|Select Footprint command. The Footprint Identification dialog box appears.
3. Specify a Tolerance to account for small variances between part instances, and for misalignment in a row of pins during the pin identification process.
4. Specify the Thru-hole Drill Layer.

5. Select Find Additional Part Instances On if you want the system to find any other instances of the part footprint you are about to define. If you do not select this option, the footprint will be added to the library, but you will not be prompted to identify any other part instances that match the footprint. If you select this option, select whether you want the system to look for other part instances on the Top Layer or Bottom Layer.
6. Click OK.
7. You are prompted to select the first pin of the part you want to identify. Click on Pin 1 for that part.
8. Select the pin that is farthest from Pin 1 (typically the opposite diagonal corner of the part). The system finds other pins automatically. If any are missed or selected by mistake, click on the pins to select/deselect them. Use the Selection Filter, if necessary.
9. Press the End key when you are finished. The Footprint Name dialog box appears.
10. The system provides a default name for the footprint you just created, using the type of device, number of pins, and the iteration of the type of footprint that it is. SMD means that it is a Surface Mount Device, THD means it is a Thru-Hole Device. For example, THD6_2 is a Thru-Hole Device with 6 pins, and it is the second footprint of that type found. You can keep the default footprint name, or assign your own, and click OK.
11. The Device Information dialog box appears. You can choose to keep the default information that the system created for the device, or specify the name and the Reference Designator Prefix you want associated with the device that you just identified. The other information (Part Number, Value, Tolerance, Description) is optional. All parts have a color-overlay assigned to them, with the pins appropriately numbered, and the centroids remained marked with a target of the same color. A bounding box is placed through the centerline of the pins. You can select a color by clicking on the color in the dialog box, or accept the system default.
12. If the system finds other pads that appear to use the same footprint as the part you just created, it highlights the pins and you are asked if they should be identified as another part. If the suggested information is correct, click Yes to identify that part. If the part information is not correct, you can specify a new device name to associate with it, the correct reference designator, or rotate the part. Then click Yes. Otherwise, click No to identify another part.

    If you click Cancel in the Device Information dialog box, then you are prompted to select the pins for a part again. If you want to abort the process entirely, press the Esc key. The process aborts, but any part information remains. You can execute this command again later, without losing any existing part information.

Related topics

[Reverse Engineering for Assembly]

Assembly|Parts|Delete

Toolbar Button: 

This function allows you to delete one or more part placements from the database. It does not alter any design data, nor device or footprint definitions. It only deletes the part identification.

Tip: This command is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

To Delete a Part Placement

1. Select the Assembly|Parts|Delete command. You are prompted to select a part to delete.
2. Select a part or parts to delete. Use the Selection Filter to aid your selections, as necessary
3. Each time you select a part for deletion, you are prompted for confirmation. If you do not want to be prompted, click the Disable Prompts button in the Status Bar.
4. When you are finished, press the Esc button.

Assembly|Parts|Query

Toolbar Button: 

This function displays information about a part, using the Selection Filter as an aid in selecting the desired part.

If more than one part exists in the location you select, the Choose Selection dialog box appears. As you click on part in the Choose Selection dialog box, each selected part is highlighted, and its attributes are displayed in the Item Properties box.

Tip: This command is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

Assembly|Parts|BOM Report

See the Documentation|Reports|BOM command for a description of this dialog box.

Assembly|Fiducials

The Fiducials commands allow you to identify and create reports for the board and part fiducials in your database.

Assembly|Fiducials|Identify

Toolbar Button: 

This function can be used to identify either board or part fiducials. If you have not identified any parts in your data using one of the Assembly|Parts|Identify Using commands, then you can only identify board fiducials.

Tip: This function is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

To Identify Fiducials

1. Select the Assembly|Fiducials|Identify command. You are prompted to select a fiducial
2. Click on the desired fiducial, and the Fiducial Identification dialog box appears.
3. Select whether the fiducial is for the board or a part.
4. Specify a name for the fiducial, if it is a board fiducial, or select the reference designator of the part it belongs to, if it is a part fiducial.
5. Click OK. The fiducial is added to the Assembly section of the Navigator. If you want to view the location of a fiducial, click on its name in the Navigator, and it is highlighted with a target.

Assembly|Fiducials|Report

See the Documentation|Reports|Fiducials command for a description of this dialog box.

Assembly|Import BOM

Toolbar Button: 

After you have reverse engineered your data for assembly, this function allows you to add additional part information from a bill of materials to the database prior to exporting.

Tip: This function is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

To Import a Bill of Materials

1. Select the Assembly|Import BOM command. The Import BOM File dialog box appears.
2. In the Filename box, specify the bill of materials file to use. Click the browse button if you need to search for the desired file.
3. Select the correct BOM File Converter for the BOM file from the Format list. These converter files are located in the BOMConv folder in your main VisualCAM folder. If a converter does not exist, see Creating a BOM File Converter for instructions.
4. Click OK, and the information from the bill of materials is added to the current footprint library. Note that if the bill of materials contains device names or part numbers that do not match those in the footprint library, they will be added as necessary.

Creating a BOM File Converter

VisualCAM provides an interactive BOM File Converter Creator that allows you to quickly create a BOM file converter file.

1. In the Use BOM File dialog box, click the New Converter button. The Create BOM File Converter dialog box appears. The name of the BOM file being imported appears at the top, and your current BOM file is displayed in the Format 1 tab.
2. Your BOM file appears in a spreadsheet, and default column names are provided at the top. These columns are not automatically matched to the BOM file; you must reorder and resize the columns to match the BOM file contents.
   - If your BOM file uses a single space as the column delimiter, select the Use Fixed Width Columns option.
   - Modal Format should be selected if your bill of materials uses one line of part information, then followed by a list of reference designators that are on their own lines (indicating that the same part information applies to each reference designator, without repeating the information on each line).
   - To move a column header, click it with your mouse and drag it to the correct location.
   - The column header width should exactly match the width of the BOM file columns. Click on the left or right edge of the column header and drag it to the appropriate location.

- If you wish to delete a column header, right-click on it and select Delete Column from the shortcut menu. Note that once you delete a column, it can always be added again later (it appears as an Add command in the right-click menu).
- If you have a column of information in your BOM file that you wish to be ignored, right-click on the column header currently above it, and select Add Skip Column from the shortcut menu. A header titled "Skip" now appears above the contents.

3. If your BOM file uses column delimiters other than single spaces, specify the appropriate Delimiter. Note that the Spaces & Tabs option assumes multiple spaces/tabs are used, as opposed to the Use Fixed Width columns option, that assumes just one space is used.
4. Select the Units of measure for your BOM file.
5. If you have any lines in the BOM file, such as headers or comments, specify in the Initial Lines To Ignore box how many lines at the beginning of the file should be ignored by the converter. Also specify, in the Comment Text box, the symbol that leads any comment lines that may be intermixed with your BOM information.
6. As you identify the columns in your BOM file, VisualCAM automatically lists recognized information in the Matched BOM entries list. All recognized entries are also colored red in the BOM Format view.
7. When you are finished, click the Save Converter button to save your ACR file in the BOMConv folder. The dialog box closes and your converter appears in the Format list of the Import BOM File dialog box.

Assembly|Export FATF

Toolbar Button: 

After you have finished reverse engineering your data, this function exports a Fabmaster ASCII Transfer Format (FATF) file for use by assembly software. For those who are familiar with the FATF format, we have included a description of the FATF option (such as :NOTRACE or :NOAUTOCOG) that corresponds with each dialog box option.

Tip: This function is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

To Export a FATF File

1. Select the Assembly|Export FATF command. The Export FATF dialog box appears.
2. Specify an Output File name. Click the browse button, if you want to search for a file/folder.
3. In the Board Thickness box, specify the thickness of your board.
4. Specify your Thru-Hole Drill Layer.
5. For the Units (Inches), select the precision of the output measurements.
6. Select Automatic Trace Connection Of Unconnected Nodes to make all traces connect to the exact center of the pads, extending any traces that do not terminate at the center of the pad. When this option is not selected, the :NOTRACE option is generated.
7. Select Automatic Center Of Gravity Calculation to generate the centroids of the silkscreen bodies (openings). This is in addition to the part centroids. When this option is not selected, the :NOAUTOCOG option is generated.
8. Select Automatic Part Rotation to have your assembly software redefine the parts in the design so that the vector components showing the direction of first pin towards the second pin are
   - positive for the X axis (0 < Xpin2 - Xpin1)
   - positive or zero for the Y axis (0 <= Ypin2 - Ypin1)

   When this option is not selected, the :NOAUTOROTATE option is generated.
9. Select Group Identical Pad Stacks to find padstacks with the same qualities and give them the same definition in the file. When this option is not selected, the :NO_PAD_STACK_GROUP option is generated.
10. In the FATF database, normally a package is created for each footprint, using the footprint name. The Define A Package For Each Device option creates a package for each device, using the device name. This results in a greater number of unique packages.
11. If you are generating the files for assembly, then you will want to include the fiducial section, and select the Export Fiducial Section option. If you are generating FATF files for assembly in-circuit test, then do not select the Export Fiducial Section option.
12. If you are generating the files for assembly in-circuit test, select All to output all viable test locations on vias and test point pads. Select Solder Mask to only output vias and test point pads that are exposed (are not covered by a mask). Select None if you do not want to generate assembly test data.
13. Click OK. The system checks for any data errors, such as whether pads exist on plated thru-hole drills, and informs you of any errors. Your FATF file is then created.

User menu

The User menu contains user-definable commands. The purpose of the User menu is to allow you to make your favorite Macros and commands as accessible and easy to use as any other VisualCAM command. Between the User menu, toolbars, and programmable mouse/function keys, the commands you use the most are truly a keystroke or mouse click away.

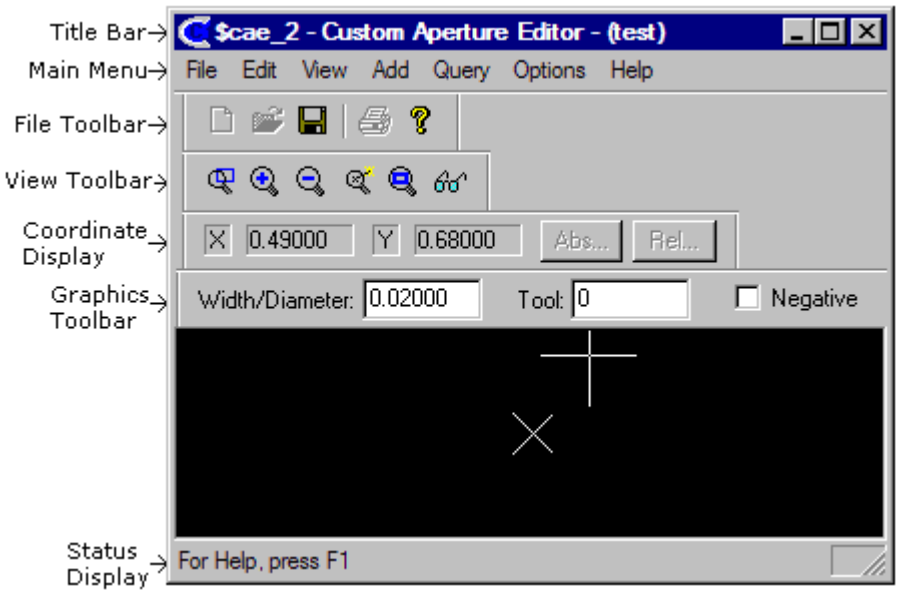
There are some commands already provided for you in the menu. For more information on how to set up your own commands, see the User Menu tab in the Options|Configure dialog box.

Custom Aperture Editor

A custom aperture is a collection of graphic drawing primitives such as circles and lines. The easiest method of creating a custom aperture is to use the Tools|Convert|To Custom command. This command allows you to window around a group of items on a layer and have them automatically converted into a custom aperture.

The Custom Aperture Editor provides a separate workspace in which to create and modify custom apertures. It is opened when you select Custom Ap/New or Custom Ap/Edit command in the Aperture Setup dialog box.

Most of the menu commands in the Custom Aperture Editor window behave in a similar fashion to those in the VisualCAM main window. Due to the nature of the Editor, all of the File menu commands except Save and Exit are unavailable.



Main Menu

The Custom Aperture Editor menus provide functions to manipulate the data in the workspace.

File Tool Bar

This toolbar contains buttons to save your custom apertures and display information about the Editor. They each have equivalent commands.

View Tool Bar

This toolbar contains buttons to manipulate the display of your data in the workspace. They each have equivalent commands in the View menu.

Coordinate Display

This toolbar shows the X and Y coordinates of your cursor in the workspace. The Abs and Rel buttons allow you to change the coordinates to Absolute or Relative. The buttons are available when you are in one of the Edit or Add function.

Graphics Tool Bar

When you add a line or circle in the workspace, you control the diameter of the circle or width of the line by entering a value in the Width/Diameter text. If your custom aperture contains drill information, the current drill tool number is shown. (Drill tool assignment is not a requirement for creating a custom aperture, unless you are creating a custom from drill layer data to create complex step and repeat patterns. 0 is displayed if no drill tool information is present.) If you would like to place the items as negative data, select the Negative option.

Status Display

This toolbar provides prompts to aid you in whatever function you are using

Custom Aperture Editor Command Reference

File|Save

Toolbar Button: 

Equivalent Hotkey: CTRL+S

The File|Save command in the Custom Aperture Editor saves the custom aperture in the workspace, and exits the Editor.

File|Exit

File|Exit closes the Custom Aperture Editor and returns you to the Aperture Setup dialog box. If you have made any edits to a custom aperture, you are prompted to save your changes prior to exiting the Editor.

Edit|Undo

Equivalent Hotkey: CTRL+Z or **U**

Edit|Undo reverses the last change made in the Custom Aperture Editor. Note that you can only undo the last change you made. Selecting the Undo command twice has no effect.

Edit|Copy

Equivalent Hotkey: CTRL+C

Use the Copy command to copy graphics in a custom aperture.

To copy one item

1. Select the Edit|Copy command.
2. Click on the item you want to copy. An outline of the item is attached to the cursor.
3. Move the mouse cursor to the desired location of the copy, and click on it.
4. Select another location for another copy by clicking on it, or right-click to select another item to copy.
5. Right-click a second time to end the command.

To copy multiple items

1. Select the Edit|Copy command.
2. To copy multiple items, you must create a selection window. Click on the desired upper-left corner of the selection window.
3. While still holding down the mouse button, move the cursor to the lower-right corner and release. All items contained entirely within the window are highlighted.
4. Click on any of the selected items to attach all of them to the cursor.
5. Move the cursor to the desired location of the copies, and click to place them.
6. Move the cursor and click again on a new location to add more copies, or right-click to select other items to copy.
7. Right-click a second time to end the command.

Edit|Move

Use the Move command to move graphics in a custom aperture.

To move one item

1. Select the Edit|Move command.
2. Click on the item you want to move. An outline of the item is attached to the cursor.
3. Move the mouse cursor to the new location and click on it.
4. Select another item to move, or right-click to end the command.

To move multiple items

1. Select the Edit|Move command.
2. To move multiple items at once, you must create a selection window. Click on the desired upper-left corner of the selection window.
3. While still holding down the mouse button, move the cursor to the lower-right corner and release. All items contained entirely within the window are highlighted.
4. Click on any of the selected items to attach all of them to the cursor.

5. Move the cursor to the new location and click to place them.
6. Select other items to move, or right-click to end the command.

Edit|Delete

Use the Delete command to delete graphics in a custom aperture.

To delete one item

1. Select the Edit|Delete command.
2. Click on the item you want to delete. The item is highlighted, and you are asked to confirm the deletion.

To delete multiple items

1. Select the Edit|Delete command.
2. To delete multiple items at once, you must create a selection window. Click on the desired upper-left corner of the selection window.
3. While still holding down the mouse button, move the cursor to the lower-right corner and release. All items contained entirely within the window are highlighted, and you are asked to confirm the deletion.

Edit|Change Drill Tools

This command is used to change the drill tool assigned to selected data. Drill tool assignment is not a requirement for creating a custom aperture, unless you are creating a custom from NC layer data to create complex step and repeat patterns.

To change drill tool assignments

1. Select the Edit|Change Drill Tools command.
2. Specify the tool number you are changing from, the tool you are changing to, and the size of the new tool.
3. Click OK, and select the item whose drill tool you wish to change.
4. To change the information for one item, click on it. The item is highlighted and you are asked to confirm the change.

   -or-

   To change the drill tool for multiple items at once, you must create a selection window. Click on the desired upper-left corner of the selection window. While still holding down the mouse button, move the cursor to the lower-right corner and release. All items in the window with the drill tool assignment you are changing from are highlighted. You are asked to confirm the change.
5. Right-click to end the function.

View|Window

Toolbar Button: 

Use the View|Window command when you want precise control over the view in the workspace. Two points are required to define a rectangle that encompasses the area that is to become the new viewing window.

If you are currently zoomed in very close to the workspace, and wish to expand your view, use the View|Zoom Out command.

To create a view window

1. Click on the upper-left corner of your desired view space. A window is attached to the mouse cursor.
2. Move the cursor and click on the lower-right corner of your desired view space. The workspace immediately zooms in on your selection.

Related topics

View|Zoom In
View|Pan

View|Zoom In

Toolbar Button: 

Equivalent Hotkey: I or **+ (plus key)**

Use the Zoom In command to quickly decrease the size of the viewing window, and see a smaller area of your design in more detail, in the workspace.

If you would like more precise control over the area you wish to view, use the View|Window command.

To zoom in

1. Select the View|Zoom In command. You are prompted to click on a center point.
2. Click on the point in the workspace that you wish to be the center point of your viewing window. The size of the current viewing window is reduced by half.

View|Zoom Out

Toolbar Button: 

Equivalent Hotkey: O or **– (minus key)**

Use the Zoom Out command to quickly increase the size of the viewing window, and see more of your design in the workspace.

To zoom out

1. Select the View|Zoom Out command. You are prompted to click on a center point.
2. Click on the point in the workspace that you wish to be the center point of your viewing window. The size of the current viewing window expands (doubles in size).

View|Pan

Toolbar Button: 

Equivalent Hotkey: P

The Pan command moves the current viewing window to a new location, which is centered on a point you specify. This command does not change the size of the viewing window.

Tip: Use Ctrl+P to toggle "Autopan" mode. In this mode, when you move your cursor at the edge of the workspace, the workspace view automatically pans in that direction.

To pan your view

1. Select the View|Pan command. You are prompted to select a new center point for your view.
2. Click on the new view window center point in the workspace.

   -or-

   Click the Abs button in the XY Bar and select the exact coordinates for the center of your new view window.

   -or-

   Click the Rel button in the XY Bar and select coordinates that are relative to the current center point of the view window.

   After you select the center point, the display immediately pans to the new location.

Related topics

View|Window

View|All

Toolbar Button: 

The View|All command adjusts the size of the viewing window to encompass the extremes of the custom aperture.

View|Redraw

Toolbar Button: 

Equivalent Hotkey: R

The Redraw command refreshes the display in the workspace.

View|Sketch

Toolbar Button: 

Equivalent Hotkey: F

This command toggles Sketch mode on/off in the Custom Aperture Editor. When sketch mode is enabled, items are shown with an outline only.

View|Toolbar

This command toggles the display of the Custom Aperture Editor File toolbar.

View|Status Bar

This command toggles the display of the Custom Aperture Editor Status Bar at the bottom of the window.

View|View Bar

This command toggles the display of the Custom Aperture Editor View toolbar

Add|Circle

Use this command to add a filled circle to your custom aperture.

To add a circle

1. Select the Add|Circle command. An outline of the circle is attached to your cursor, and you are prompted to select the location for it.
2. If you would like to change the diameter of the circle, enter the value in the Width/Diameter text box in the Graphics Toolbar. If you would like to place the circle as negative data, select the Negative option in the Graphics Toolbar.
3. Click on the location of the circle. You can click on more locations, as desired, or right-click to end the command.

Add|Line

Use this command to add a line to your custom aperture.

To create line segments

1. Select the Add|Line command. If you would like to change the width of the line, enter the value in the Width/Diameter text box in the Graphics Toolbar. If you would like to place the line as negative data, select the Negative option in the Graphics Toolbar.
2. Click on the location of the first end point of your line. A representation of the line is now attached to your cursor.
3. Move your cursor to the location of the next vertex in your line, and click on it. You may continue to add vertices to your line.
4. After you have placed the end-point of your line, right-click to end the command.

To create an arc

1. Select the Add|Line command. If you would like to change the width of the line, enter the value in the Width/Diameter text box in the Graphics Toolbar. If you would like to place the line as negative data, select the Negative option in the Graphics Toolbar.
2. Click on the location of the first end point of your arc.
3. Press the A key to switch the Add Line function to Add Arc.
4. Click on the second end point of your arc.
5. You can now either click on the third point on the circumference of the arc, or press the 9 key to create a 90 degree arc.
6. Right-click to end the command.

Query|Item

This command provides you with information about the lines, circles, and arcs that comprise your custom aperture.

To query an item

1. Select Query|Item, and you are prompted to select an item in your custom aperture.
2. Click on any circle, line, or arc in your custom aperture. The item is highlighted, and the results of your query are displayed in the Status Bar at the bottom of the Editor window. You are told what the item is, its width or diameter, whether it is positive or negative data, and the coordinates of its end points. If there are multiple objects at the same location, you can continue to click on that location to cycle through the objects.
3. When you are finished, press the Esc key.

Options|Grid

Equivalent Hotkey: CTRL+G

You can control the grid in the Custom Aperture Editor workspace with this command.

To change the grid settings

1. Select the Options|Grid command. The Grid dialog box appears
2. To make your mouse cursor snap to the grid, select the Snap option.
3. Select the Display option to make the grid visible in the workspace.

4. The Size value controls the distance between points on the grid. By default, the value is Metric. If you would like the value to be in inches, see the Options|Metric command.
5. When you are finished, click the OK button.

Options|Metric

By default, the Custom Aperture Editor units of measure are Metric. If you would like to change the units of measure to Inches, unselect the Metric command.

The units of measure are always displayed in the right side of the Status Bar, at the bottom of the window.

File Format Technical Reference

Aperture List Files

Aperture lists are stored as simple ASCII files. Each line defines one D-code, and there are eleven fields in each line of the file that describe the D-code's properties. Exporting an Aperture List is only required if you are exporting your database as 274-D Gerber data.

VisualCAM supports several intrinsic apertures. While the support of these apertures is reflected in the Aperture List file, as described below, it is highly recommended that you instead export your data in the 274-X or ODB++ file format, as not all 274-D equipment will properly support these apertures as intrinsics.

The fields of a VisualCAM aperture list consist of the following:

Field	Possible Values
D-Code	10 - 9990
Shape	Round, Square, Butterfly, Oblong, Octagon, Diamond, Triangle, Bullet, Ellipse, Hexagon, Donut, Thermal, Moiré, or Rectangle. Custom apertures are preceded by "%".
Width	0.0 - 9.9999
Height	0.0 - 9.9999 When referring to Donuts or Thermals, this field represents the diameter of the inner hole. When referring to Targets, this field refers to the diameter of the inner ring of the Target.
Type	SMT, Thruhole or Thermal
Tool	0 - 999 *Not used after Version 8.*
Tool Size	0.0 - 9.9999 *Not used after Version 8.*
Legend	10 - 4095 *Not used after Version 8.*
R90	10 - 4095 The D-code to substitute for this D-code when rotating 90 or 270 degrees. This field exists only for compatibility with older versions of GerbTool as newer versions perform the D-code substitutions automatically (a "0" will appear in that case).
Angle	The angle of rotation for the aperture. A positive number indicates counter-clockwise rotation. A negative number indicates clockwise rotation.
Properties	The properties of each intrinsic aperture depend upon its shape. All sizes, widths, lengths, etc. in the Properties field are in VisualCAM database units. To convert the values to inches, divide them by 2540000. All angles are in degrees. Styles and shapes are ASCII strings exactly like those in the Aperture Setup dialog box, except for the Rectangle corner style, as indicated below. **Round Size** **Square Size** **Butterfly Size Shape** **Oblong Xsize Ysize** **Octagon Xsize Ysize** **Diamond Xsize Ysize** **Triangle Xsize Ysize** **Bullet Xsize Ysize** **Ellipse Xsize Ysize** **Custom Xsize Ysize** **Hexagon Xsize Ysize CornerSize** **Donut Size InnerSize Shape** **Thermal Xsize Ysize AirGap AirGapEndStyle OuterShape InnerShape SpokeCount SpokeWidth SpokeAngle** **Moire RingCount RingWidth RingGap LineWidth LineLength** **Rectangle Xsize Ysize CornerStyle CornerFlags CornerRadius** CornerStyle: 1=Square, 2=Round, 3=Chamfer CornerFlags (the corners that the styles apply to) is a composite value made from summing the following (value is 0 if no flags are set): 1=LowerLeft, 2=UpperLeft, 4=LowerRight, 8=UpperRight.

All fields are separated by white space. Lines that begin with "#" are treated as comments. Although the author and data comments are not required, they are generally included as an aid for other users. The header of a VisualCAM aperture list may contain a format line preceded by "%". This line contains either "IMPERIAL" or "METRIC" followed by a version number. If IMPERIAL is specified, all sizes are in inches. If METRIC is specified, they are in millimeters. If no format line is provided, IMPERIAL is assumed. The version number is for documentation purposes only.

Following is an example of a VisualCAM aperture list:

```
# Format, Version
%IMPERIAL, 15.0
#
# Units: Inches
# Author: VisualCAM
# Date:   Wed Jul 30 10:04:45 2006
#
#    Shape     Width   Height    Type  Tool   Tool Size   Legend   R90 Angle Properties
#
D10  Round     0.004000 0.004000   ThruHole 0 0.0 D0      0 0 27b0
D11  Square    0.006000 0.006000   ThruHole 0 0.0 D0      0 25 3b88
D12  Oblong    0.010000 0.007500   ThruHole 0 0.0 D0      0 0 6338 4a6a
D13  Bullet    0.018000 0.018000   ThruHole 0 0.0 D0      0 0 b298 b298
D14  Moire     0.025000 0.025000   ThruHole 0 0.0 D0      0 0 2 27b0 15d4 27b0 f80c
D15  Ellipse   0.050000 0.050000   ThruHole 0 0.0 D0      0 0 1f018 1f018
D16  %custom   0.100000 0.080000   SMT 0 0.0 D0      0 0 3e030 319c0
D17  Hexagon   0.150000 0.129900   ThruHole 0 0.0 D0      0 0 5d048 508da 17412
D18  Butterfly 0.015000 0.015000   SMT   0 0.0 D0      0 0 94d4 Round
D19  Triangle  0.030000 0.040000   SMT   0 0.0 D0      0 0 129a8 18ce0
D20  Rectangle 0.040000 0.015000   SMT   0 0.0 D0      0 0 18ce0 94d4 2 f 9ec
D21  Octagon   0.040000 0.040000   SMT   0 0.0 D0      0 0 18ce0 18ce0 7416
D22  Donut     0.070000 0.070000   SMT   0 0.0 D0      0 0 2b688 208e6 Round
D23  Diamond   0.076000 0.025000   SMT   0 0.0 D0      0 0 2f210 f80c
D24  Thermal   0.044000 0.044000   Thermal 0 0.0 D0      0 0 1b490 1b490 4f60 Parallel Round
Round 4 4f60 afc8
```

Warning about 274-D: Because 274-D does not have the notion of a "custom aperture", if you export an aperture list that contains custom apertures, the customs are listed but not completely defined (only their size is indicated). Custom apertures can be saved to library files for future use. If you re-import an aperture list that contains custom apertures, they are defined as "moires". You will then have to use the Custom Ap button in the Aperture Setup dialog box to reload or define your custom apertures. Use of 274-X will avoid these issues, because the custom aperture definitions are contained within the 274-X data.

ACR Files

An Automatic Conversion Rule (ACR) file is an ASCII file used to describe a particular aperture list, tool list, Bill of Materials (BOM) or centroid file format, using simple conversion language statements. In addition to providing the ability to convert most popular CAD and photo-plotter aperture lists directly into the popular VisualCAM format, you can also create your own ACR files for specialty, proprietary or otherwise unsupported aperture, centroid, or tool list formats.

VisualCAM's GUI-driven ACR File Creator allows you to create an ACR file for any aperture list, centroid file, BOM, or tool list quickly and easily. You can also manually create a file using a separate text editor, though using the ACR File Creator is recommended as the quickest and easiest method. In either case, once the ACR file is complete, VisualCAM will then be able to read the aperture list, centroid file, BOM, or tool list automatically.

How an ACR File Works

When importing an aperture list, the ACR describes the format of the original aperture list. Once the ACR file has been read in, VisualCAM can then convert the associated apertures in the aperture list into the proper shapes and sizes in the workspace, so that your Gerber data are correctly displayed.

An ACR file is similarly used when importing a tool list. It describes the format of the list, so that VisualCAM can add the correct tool information to the NC tool table, and display your NC data correctly.

When reverse engineering part information for assembly, an ACR file is used to import a centroid file that can be used to identify parts, or a bill of materials that contains additional part information that you may wish to add to the database prior to exporting.

For the purposes of this tutorial, we will focus on the most common types of ACR file, which are used to import aperture lists and tool lists.

Regular Expression Syntax

ACR files use a regular expression syntax to describe lines that are to be processed by the converter. A regular expression is simply a way of describing a certain pattern of text in a well defined fashion.

What follows is a description of the regular expression syntax an aperture list ACR file uses. We will first start out with the simplest types of patterns that are supported, and then progress to the more advanced ones. The simplest expression is the character. A character in the rule file will match the corresponding character in the aperture list. For example, the rule:

```
D$dcode
```

will match the line

```
D10
```

in an aperture list, but not the line

```
DCODE10
```

due to the extra characters "CODE". Some characters have special meaning in a rule. These characters include '. | + * ( ) [ ] ? $ ^ \'. It is unlikely that you will encounter any of these characters when creating an aperture converter of your own, but if you do need to use one of them, you can precede the character with a backslash (\). For instance, the string:

```
\(D$dcode\)
```

matches

```
(D10)
```

Quite often you will have a variable number of characters to match. This occurs most often with spaces. Patterns of this sort can be matched by using the characters "+" and "*". A character followed by * matches 0 or more instances of that character. A character followed by + matches 1 or more instances of that character. For example, the string:

```
ab+a
```

matches all of the following:

```
aba
abba
abbba
abbbbba
```

while the string:

```
ab*
```

matches all of those lines plus the line:

```
aa
```

Matching Order

Sometimes you will encounter instances where a line in an aperture list will match multiple FORMAT lines in your converter. If this happens, the converter will use the first FORMAT line, matched. For example, to match the following 2 lines:

```
10 THERMAL 0.25 0.15 (normal thermal)
11 THERMAL 0.25 0.15 45.0 (45 degree thermal)
```

You would want to use the following 2 lines in this exact order:

```
FORMAT_THERM45:$dcode THERMAL +$xsize +$ysize +45.0 (a)
FORMAT_THERMAL:$dcode THERMAL +$xsize +$ysize (b)
```

If you were to have the order of these lines reversed, line (b) would match both of the original aperture list lines, and the converter would produce two non-rotated thermals.

Creating an Aperture or Tool List ACR File

VisualCAM's GUI-driven ACR File Creator allows you to create an ACR file for any aperture list, centroid file, BOM, or tool list quickly and easily. You can also manually create a file using a separate text editor. This topic will assist you in the latter case, specifically for creating an ACR file for aperture lists or tool lists.

An ACR file contains two types of statements. The first type of statement describes the environment such as the expected file extension, metric mode, number of header lines to skip etc. The second type of statement is the actual rule statement. Rules are the statements that will be used to match incoming aperture list entries to a corresponding VisualCAM aperture shape.

The following is a description of each environment statement and the expected parameters if any:

NAME

Syntax: **NAME converter_name**

Parameters: **converter_name**

The name of the ACR file should be a single word.

Description: This statement will place the parameter in the header of the resulting aperture or tool list.

Example: The following example sets the name of the converter to "Allegro".

```
NAME Allegro
```

VERSION

Syntax: **VERSION version_number**

Parameters: **version_number**

The version number of the ACR file. The version number should be a single decimal number.

Description: This statement will place the parameter in the header of the resulting aperture or tool list.

Example: The following example sets the version number of the converter to "6".

```
VERSION 6
```

HEADER

Syntax: **HEADER lines_to_skip**

Parameters: **lines_to_skip**

The number of lines to skip in the header of the aperture/tool list.

Description: If this line is present, the number of lines specified will be skipped from the header of the aperture/tool list file you are attempting to convert. This is used to quickly bypass information at the top of a file that you know does not contain any aperture/tool information.

Example: The following example instructs VisualCAM to skip the first twenty lines of the list.

```
HEADER 20
```

SKIP

Syntax: **SKIP skip_string**

Parameters: **skip_string**

A text string to mark text to be skipped.

Description: If this line is present, all lines in the list that start with the given character string will be ignored.

Example: The following example will allow VisualCAM to skip over lines that begin with "#".

```
SKIP #
```

#

Syntax: **# any_text**

Parameters: **any_text**

The body of a comment.

Description: This symbol leads comments in an ACR file.

Example: The following example shows a typical comment.

```
# Created By Joe Designer
```

DEFAULT_UNITS

Syntax: **DEFAULT_UNITS mode**

Parameters: **mode**

One of "$$INCH", "$$MIL" or "$$MM"

Description: If given, will cause the values read in to be interpreted as Inches, Mils or Millimeters, depending on the value used.

Example: The following example sets the units mode to metric.

```
DEFAULT_UNITS $$MM
```

CUSTOM

Syntax: **CUSTOM yesno**

Parameters: **yesno**

Either "$$YES" or "$$NO".

Description: If set to $$YES, VisualCAM will attempt to create custom aperture names whenever possible. Otherwise a Diamond shape will be substituted. VisualCAM will add only the custom aperture names in the aperture list, not create the custom apertures themselves.

Example: The following example sets the creation of custom apertures to off.

```
CUSTOM $$NO
```

EXTENSION

Syntax: **EXTENSION extension**

Parameters: **extension**

The default aperture list extension.

Description: The default file extension of the lists you will be converting with this rule file. If the value is entered here, you will not need to enter it when specifying the list for conversion.

Example: The following example sets default aperture list extension of "mya".

```
EXTENSION mya
```

DEBUG

Syntax: **DEBUG mode**

Parameters: **mode**

A value of 0, 1 or 2.

Description: Enables debugging information to be output into the converter log file. If the value of 0 is used, no debug information will be output. If 1 is used, VisualCAM will output debug information while parsing the ACR file, and if the value is set to 2, debug information will be output while converting the file itself. This line is for advanced users only and should either not be included or be set to 0 for normal converter operation.

Example: The following example sets the current debug mode to 2.

```
DEBUG 2
```

XTENSION

Syntax: **XTENSION dll_filename**

Parameters: **dll_filename**

The name of a .dll that you supply.

Description: If present, causes the converter to look for the specified .dll file to help in converting the aperture lists. Please contact WISE Software Technical Support Services for more information on developing DLL's to assist in converting complex aperture list formats.

Example: The following example specifies a user supplied .dll.

```
XTENSION myapfmt.dll
```

DCODE

Syntax: **DCODE mode**

Parameters: **mode**

One of "$$ONLINE", "$$SEQUENTIAL" or "$$GERBER_ORDER".

Description: This line controls how D-code values will be derived. If set to $$ONLINE (the default) the codes read on each line will be used. If $$SEQUENTIAL is used, lines that match the rules given will be assigned sequential numbers. Some aperture lists have their D-codes arranged in a special non-sequential order used in certain Gerber photoplotters. This order will be used if $$GERBER_ORDER is set.

Example: The following example sets the D-code mode to sequential.

```
DCODE $$SEQUENTIAL
```

The following are descriptions of each rule statement and the expected parameters, if any:

FORMAT_shape

Syntax: **FORMAT_shape rule**

Parameters: **shape**

The possible shapes are: ROUND, SQUARE, RECT, OBLONG, DONUT, DIAMOND, OCTAGON, THERMAL, THERM45, TARGET, CUSTOM. Please note that this parameter should be combined with the FORMAT_ prefix to form a single word such as FORMAT_ROUND.

rule

A rule for matching apertures that are to be mapped to a VisualCAM "shape" aperture.

Description: If the rule matches a line in the aperture list being converted, that line will be converted into a VisualCAM "shape" aperture.

Example: The following example will match the line "JUNK D10 0.060 0.060 ROUND".

```
FORMAT_ROUND $skip +D$dcode $xsize $ysize ROUND
```

FORMAT_UNITS

Syntax: **FORMAT_UNITS rule**

Parameters: **rule**

A rule for matching a line in the aperture list that specifies the format of the file.

Description: A line matching this is used to determine the format of the aperture list. This statement allows the aperture list itself to override a previous UNITS statement.

Example: The following example will match the line "FORMAT MM".

```
FORMAT_UNITS $skip $units
```

FORMAT_SPECIAL

Syntax: **FORMAT_SPECIAL rule**

Parameters: **rule**

A rule for matching lines for use by an XTENSION .dll.

Description: Does not produce a VisualCAM D-code line. It is used for special processing by an XTENSION-specified .dll.

Example: The following example will match the line "SQR D10 0.060 0.060".

```
FORMAT_SPECIAL SQR +D$dcode $xsize $ysize
```

FORMAT_plating

Syntax: **FORMAT_plating rule**

Parameters: **plating**

PTH indicates a plated tool, NPTH indicates a non-plated tool.

rule

A rule for matching tools that are to be matched to plated or non-plated tools.

Description: If the rule matches a line in the tool list being converted, that line will be converted into a plated or non-plated tool.

Example: The following example will match the line "T1 PLATED 0.060"

```
FORMAT_PTH $tool PLATED +$xsize
```

When constructing rules to match apertures or tool lists, there are special key words that you place in the rule that will cause VisualCAM to assign the values contained in the desired fields to the corresponding VisualCAM aperture list or tool table fields. These keywords are as follows:

Keyword	Meaning
$dcode	Assign to D-code.
$xsize	Assign to X size.
$od	Assign to X size.
$ysize	Assign to Y size.
$id	Assign to Y size.
$rot	Assign to rotation.
$skip	Skip this field.
$custom	Use this field to make a custom aperture.
$units	Used to determine the units of measure.
$tool	Assign to tool number.
$plunge	Assign to tool plunge rate.
$retract	Assign to tool retract rate.
$feed	Assign to tool feed rate.
$rpm	Assign to tool rotation speed.
$depthoffset	Assign to tool depth offset.
$maxhits	Assign to maximum number of hits for a tool.
$linktool	Assign to replacement tool.

Sample ACR file

```
# Aperture converter for Mentor
NAME Mentor
VERSION 2.0
EXTENSION rpt
# handle swapped X/Y columns
XTENSION mentor.dll
CUSTOM $$NO
DEFAULT_UNITS $$INCH
HEADER 1
FORMAT_ROUND:$skip +circle +$skip +$xsize +$ysize +$rot +false +false +$dcode
FORMAT_THERMAL:$skip +circle +$skip +$xsize +$ysize +$rot +false +true +$dcode
FORMAT_RECT:$skip +rectangle +$skip +$xsize +$ysize +$rot +false +false +$dcode
FORMAT_ROUND:$skip +circle +$skip, $skip +$xsize +$ysize +$rot +false +false +$dcode
FORMAT_THERMAL:$skip +circle +$skip, $skip +$xsize +$ysize +$rot +false +true +$dcode
FORMAT_RECT:$skip +rectangle +$skip, $skip +$xsize +$ysize +$rot +false +false +$dcode
FORMAT_SPECIAL:Position +Shape
# Mentor now has multiple formats
FORMAT_THERMAL: +$skip +$dcode +circle +$skip +power +$xsize +$ysize
FORMAT_ROUND: +$skip +$dcode +circle +$skip +$xsize +$ysize
FORMAT_ROUND: +$skip +$dcode +circle +$skip +$xsize
FORMAT_RECT: +$skip +$dcode +rectangle +$skip +$xsize +$ysize
FORMAT_RECT: +$skip +$dcode +rectangular +$skip +$xsize +$ysize
FORMAT_SPECIAL: Aperture Position
FORMAT_SPECIAL:Aperture Position
FORMAT_SPECIAL:$skip Position
FORMAT_SPECIAL: +Ap.Pos.
```

Text Font Files

VisualCAM uses a font file containing a list of X:Y coordinate pairs that constitute the "strokes" required to display each character inserted by the Add|Text command. You may have more than one font file, but VisualCAM will always read the stroke.fnt file at startup. To use a different font file, such as the fancy.fnt file that is provided in the Fonts folder, simply rename the current stroke.fnt to some other meaningful name and then rename the desired font file to "stroke.fnt".

VisualCAM allows you to edit existing fonts and create new fonts. It is faster and easier to modify the default fonts than it is to create a new font. Therefore, even if you want to create a font that is totally different than the GerbTool-Stroke font, it is recommended that you follow the instructions below.

Editing the GerbTool-Stroke Font

1. From the Windows desktop, select Start>Run and execute the following command line from the Run dialog box:

   On Windows Vista **"C:\ProgramData\WISE Software Solutions\VisualCAM 2006 15.1\Fonts\F2G.EXE" ../stroke.fnt**

   On all other Windows operating systems **"C:\Documents and Settings\All Users\Application Data\WISE Software Solutions\VisualCAM 2006 15.1\Fonts\F2G.EXE" ../stroke.fnt**

   This command line assumes that you installed GerbTool on the C: drive. If not, alter the file path as necessary.
2. This creates an individual Gerber file for each character in the font file, and places all of them in the Fonts folder.
3. Start VisualCAM.
4. Select the File|Open command and open the fonts.vcam file (in the Fonts folder). You will be warned that the file was created with an earlier version of GerbTool. Just click the Yes button to load it.
5. Each character is loaded as an individual layer. The Film Box is set to a 7mil square which each character must remain within. You may draw any shape you desire as long as you stay in or on the Film Box and you do not try to add flashes.
6. When you are finished, select the File|Export|Gerber command. The Export Gerber dialog box appears.
7. Keep the Fonts folder as the destination folder, and leave all the .gbr files selected with their default names. Click the Data Format button, and the Gerber Export Data Format dialog box appears.
8. The G2F program, which you will use to convert the Gerber files to a .fnt file, requires the Gerber files to be in a very specific format. Select RS274D as the Dialect and 2.3 as the m.n setting (you will need to type this in manually, it is not offered in the selection list). The Coordinate Mode should be set at Absolute, Zero Suppression at None, 'G' Commands should be set at Include, and Character Set should be ASCII. No not select any of the other options.
9. Click OK to return to the Export Gerber dialog box.
10. Click OK to export the files. You will be prompted whether you want to replace the existing .gbr files. Click the Yes To All button to replace all of them.
11. Select File|Exit to exit VisualCAM. When you are asked if you wish to save any changes to FONTS.GTD, click the No button.
12. From the Windows desktop, select Start>Run and execute the following command line from the Run dialog box:

    On Windows Vista **"C:\ProgramData\WISE Software Solutions\VisualCAM 2006 15.1\Fonts\G2F.EXE" newfile.fnt**

    On all other Windows operating systems **"C:\Documents and Settings\All Users\Application**

    Data\WISE Software Solutions\VisualCAM 2006 15.1\Fonts\G2F.EXE" newfile.fnt

    In the above example a new font file called "newfile.fnt" is created in the Fonts folder. Please note that this does not purge the individual Gerber character files from the Fonts folder. You may do this manually, if desired.
13. To use the font file you created, rename the existing stroke.fnt file in the VisualCAM 2006 15.1 folder. Then copy the "newfile.fnt" file to the VisualCAM 2006 15.1 folder, and rename it to "stroke.fnt".
14. You may now start VisualCAM and use the modified font to create text.

Sample VisualCAM Netlist (.nl) File

The following is a sample netlist file created by the Tools|Netlist|Save command. This net was saved using net numbers instead of user data for labeling. The number pairs are the coordinates of the net points.

```
NET 1
   825,4600 2300,5225 2300,5325 2700,4750
NET 2
   875,4700 975,4700
NET 3
   900,4500 1300,1600 1300,4200 1350,4500
NET 4
   900,4600 2400,5175 2400,5275 2400,5375
NET 5
   975,3375 1700,3375 1700,3475
NET 6
   975,4000 1100,3900 1600,4200
NET 7
   3300,4000 3400,3375 3400,3700 3400,3800 3400,3900 3500,3675 3500,4050
NET 8
   975,4800 1300,5000
NET 9
   1000,4400 1000,4500 1100,4200 1200,1600
NET 10
   1100,3375 1500,2725 2125,2625
NET 11
   1200,2325 1400,1000 1400,2725 2725,750
NET 12
   1200,3075 2275,3050 2275,3150
NET 13
   1300,3375 2125,2525
NET 14
   1400,1300 1600,1300 1600,2725 2525,750
```

Color List File

When starting up, VisualCAM loads a color list file, named "color.rgb". VisualCAM reads first the available colors from a RGB color/name pair list and second, a list of current choice colors. The current choice colors specified will be those presented whenever you select colors within VisualCAM (i.e. flash and draw colors). You may modify both the color/name pair list and the color choice list to meet you needs.

```
# maximum 1024 colors available...
[RGB Color/Name pairs]
128 0 0  vga16red
0 128 128  vga16cyan
0 128 0  vga16green
245 245 245  WhiteSmoke
  .
  .
  .
  .
255 250 240  FloralWhite
253 245 230  OldLace
250 240 230  linen
250 235 215  AntiqueWhite

# maximum 32 current choice colors...
[Choice Colors]
blue
vga16green
white
black
coral
 .
 .
 .
 .
SteelBlue
SaddleBrown
DarkSalmon
DarkOrange
DeepPink
```

Glossary

A

absolute coordinates: The actual X:Y coordinate value, based on a 0:0 origin.

acid trap: An area where etching solution accumulates but does not flow out during manufacturing. This causes over-etching.

active layer: The current layer that you are working on in the workspace. The currently active layer is displayed in the Layer Bar.

antenna: A net that does not end on a pad.

aperture conversion rule file: An ASCII file used to describe a particular aperture list format, using simple conversion language statements. GerbTool uses this ACR file when you import aperture lists, so that your design data appears with the correct D-codes.

aperture list: A list of Gerber aperture definitions (D-code number, size, and shape). Separate aperture list files are used with 274-D data, while 274-X has the aperture list imbedded in the Gerber file.

aperture report: Details which D-codes, along with their definitions, are being used on a per-layer basis. The report includes use counts for both flashes and draws.

ASCII: American Standard Code for Information Interchange. This standard relates alphabet characters to specific code numbers.

B

blind via: A via that is exposed on either the top or bottom layer, and passes through an internal layer, but does not pass completely through the board. (When looking at a finished board, you can see where the via starts on one side, but not where it ends inside.)

border: A continuous outline of the board, which is placed on a layer by itself. This layer is tagged with the "border" type.

breakout: Pad and trace separation during manufacturing.

buried via: A via that passes through certain internal layers only. It is not exposed on the top or bottom layers. (The via is "buried" inside the layers.)

C

CAD: Computer-Aided Design

CAM: Computer-Aided Manufacturing

compensation: Compensation is used to offset the mill path from the centerline. This is most often used when you convert a border into a mill path, and you wish to offset the path so that the mill tool cuts outside of the border.

composite: A composite a single layer (and hence, a single Gerber file), which is made up of a set of positive (dark) and negative (clear) layers. They are used to route traces on power planes.

copper area: The amount of copper that is used in your design.

copper sliver: An area of copper that is so narrow that it is likely to flake off during manufacturing.

custom aperture: Special user-created shapes that are normally used as flashes. They can be created or modified in the Custom Aperture Editor.

D

D-code: Draft code. Aperture shapes, which are used to create your designs, are referenced by D-code numbers in Gerber data.

DFM: Design For Manufacture

dimensioning: Displaying the distance between two items in the design, using lines, arrows, and text. Dimension information is placed on a drawing layer, separate from normal layer data.

draw: Instances of a D-code that occur sequentially from one X:Y location to another, to form a line segment.

drawing layer: A layer used to add information that you do not wish to place on your design layers. This includes redline information, mechanical drawings and diagrams complete with "intelligent" drawing primitives which automatically update themselves.

DRC: Design Rule Checking

drill path: The order in which the drill machine creates drill hits.

drill tool report: Details which drill tools, along with their definitions, are being used on all drill layers.

DXF: Data Exchange Format, a file format introduced by AutoCAD.

E

EBCDIC: Extended Binary Coded Decimal Interchange Code

EIA: Electronics Industry Association

endcap: The DXF data format draws lines with square shapes. To maintain a round end to a line, a round shape is added to the line endpoints.

F

FATF: Fabmaster ASCII Transfer Format

film box: A continuous line that represents the film size.

fine pitch pads: A row (group) of pads whose center-to-center or edge spacing is very small (usually less than 10 mils).

flash: An instance of a D-code at a single X:Y location.

G

Gerber: A file format that contains a design's graphical data. Each file contains a single layer.

grid: Evenly spaced points that fill the workspace, which are used as a visual aid for locating coordinates and measuring the distance between objects.

grid snap: The cross-hair cursor, attached to the mouse cursor, automatically jumps to the nearest grid point in the workspace.

H

hole chart: A hole chart is a legend that shows the symbol being used to represent a particular set of drill hits, the size of the drill that is associated with those hits, whether they are plated or unplated, and how many hits there are of each drill.

hotkey: A key on your keyboard that executes VisualCAM functions. Also referred to as a "nested command," hotkeys may be used when another command is active, without interrupting the active command.

HPGL: Hewlett Packard Graphic Language

I

incremental coordinates: Each X:Y coordinate is a displacement from the previous coordinate.

isolated pads: Pads that do not have a trace connected to them.

isolated thermal: A thermal that is insolated from the rest of a negative plane, due to over-etching of surrounding items.

L

layer set: A standard layer set is a user-defined group of layers, whose visibility can be altered simultaneously. Blind/buried layer sets are used to define the relationship of drill layers to circuit layers, which contain blind & buried vias, prior to generating a netlist. MCM/LTCC stackups define the sequence of electrical and insulating layers for MCM or LTCC boards.

legend D-code: The D-code used when displaying drill, mill, test point, or other manufacturing information in the workspace.

LTCC: Low Temperature Co-fired Ceramic

M

m.n: The coordinate format variable that specifies the location of the implied decimal point. A format of 2.3 specifies that there are 2 digits before the decimal point, and 3 digits after the decimal point. Using this example, the number 12345 in a Gerber file would represent 12.345.

macro: VisualCAM functions that the user can define in the Macro Developer, and automatically perform in a sequence. Macros can be saved to a file, and used to streamline processes.

mask sliver: An area where the resist is so narrow that it will likely flake off, float, and redeposit itself in an area that might be soldered later.

MCM: Microchip Module

MRC: Manufacturing Rule Checking

MRU: Most Recently Used

N

NC: Numerical Control. Refers to files that control drilling and milling machines.

NC tool list: A list of drill and mill tools, and their properties.

NC tool report: Details of which mill and drill tools, along with their definitions, are used on all NC layers.

net number: A unique identifier for a net.

netlist: A file containing groups of X:Y locations for pads that are connected by traces.

NPTH: Non-Plated Through-Hole

O

open: A break in a net.

origin: An X:Y point (usually 0:0) used as the basis for the location/coordinates of all the data in your design.

P

panelize: Placing multiple copies of a PCB on one panel for manufacturing. This allows several boards to be manufactured at once, reducing time and costs.

PCB: Printed Circuit Board

pilot hole: A preliminary, small hole that is drilled at the plunge point of a mill path or at the point of a larger drill hit. It is used as a guide to aid the larger tool.

pin hole: A void in an area of solid copper that can cause acid to pool and over-etch the surrounding copper, or cause resist flakes.

plunge point: The location where the mill machine inserts the tool and begins milling.

point: An X:Y location in the workspace.

polarity: Positive polarity refers to dark, or visible database items. Negative polarity refers to data which clears away (erases) visible database items, so that the background color appears instead.

polyline: A contiguous line made of multiple drawn segments.

PTH: Plated Through-Hole

R

redline: Information or comments about design data, stored on a drawing layer (separate from normal layer data).

relative coordinates: X:Y coordinates are given relative to a selected point.

resist sliver: A small area of resistive material that has a surface area too small to adhere to the board, and therefore flakes or peels off. Where acid traps are open at one end, resist slivers can be fully surrounded by copper.

retract point: The location where the mill machine removes the tool after milling.

S

short: An unwanted connection between two or more nets.

sketch mode: Only the outline of items is displayed in the workspace.

slot: A drill tool is used to create a series of hits, in a straight line, to make a path that appears as though it was created by a mill tool.

SMD: Surface Mount Device. A component that mounts on only one side of a board. SMD pads mount the component to the board without using through holes.

SMT: Surface Mounting Technology

solder bridge: An unwanted connection between a pad and an adjacent trace or other conductive object. It occurs when the opening for a pad on a mask layer is oversized too much, and exposes the adjacent object. The copper for the pad may get too close during fabrication, causing the accidental connection.

starved thermal: A thermal whose connection to the negative plane is constricted by adjacent data.

step & repeat: Gerber codes which allow single images to be arrayed in a panel without creating new copies of the original; this helps reduce file sizes.

stick mode: All drawn data are displayed in the workspace as a single line.

T

tab: Short "breaks" in the mill path that are used when you are milling completely around a board. In order to keep the board from literally dropping out of the material, the machine stops at certain points, extracts the mill tool, moves a short distance, and continues the mill path.

teardrop: Adding additional material to smooth the transition from trace to pad, strengthening the trace-pad junction and preventing drill breakout at the junction.

test point: A point on a net, used by a testing machine, to verify net connectivity.

thieving: A solid border of copper placed around the images in a panel, intended to prevent over-etching of a panel layer during the manufacturing process (also known as galvanic venting or robber bars). Unlike venting, thieving does not fill the areas between images.

U

user data: Information attached to a database item, which is saved in your database (but does not add any graphics to your design). User data can be used to associate reference designators, pin numbers, or net names to each pad, or it can simply be information that you wish to pass on to others who view the design.

V

venting: A cross-hatch or dot pattern added between panel images, to the inner layers.

virtual panelization: Creating a manufacturing panel using step & repeat codes.

Index

274-D files 47
274-X files 47

A

Abs button 35
absolute coordinates 35, 47
acid traps 164
ACR file 47, 235
  creating 82
  sample 240
active break tab 6
active D-code 6
active layer 6, 36
active tool 6
AD command 47
add
  apertures 57, 140
  arcs 129, 131
  arrays 131
  blind/buried layer sets 145
  circles 130
  composites 13, 144
  construction lines 160
  custom apertures 10
  dimensions 157
  draws 129
  drill hits 134
  drill slots 134
  drill tools 146
  drilled circles 135
  drilled text 136
  fabrication note balloons 162
  flashes 129
  layer sets 145
  layers 139
  layers to composites 13, 144
  mill paths 134, 196
  mill tools 146
  milled circles 135, 136
  NC tools 146
  operator messages 138
  optional stops 138
  polygons 132
  rectangles 130
  redline arrows 155
  redline lines 155
  redline sketches 156
  redline text 153, 154
  teardrops 194
  user data 106
  vertices 111, 130
Add Arrow command 155
Add Balloon Text command 154
Add command 111, 127, 201
Add Dimension command 157
Add Hole Chart command 160
Add Line command 155, 160
Add Note Balloon command 162
Add Text command 153
Add To command 105
adjacency 202
Advanced command 182
Aerial Bar 35
align
  drill hits 116
  layers 116
  pads 116
Align Layers command 116
All (Fit) command 124
All command 230
Analysis 22
Analysis command 202
analysis errors 22
  finding 64, 164
  fixing 22
  highlighting in workspace 22
  printing reports 22
  searching for 22
  viewing 22
analysis pass 22, 164
  printing reports 22
  viewing information 22
analysis run 22, 164
  deleting 22
  printing reports 22
  renaming 22
  viewing information 22
aperture conversion rule files 47, 57
  sample 240
aperture list
  exporting 95
  files 47, 57, 233
  importing 79, 82
  viewing 140
Aperture List command 82, 95
aperture macro 47
apertures 6, 8, 47, 57
  adding 8, 57, 140
  custom 140
  deleting 8, 57, 140
  editing 8, 57, 140
  exporting 57
  importing 57
  loading from library files 140
  rotating 140
  use report 151
Apertures command 140, 151
Apply External Net Names command 65, 188
Arc 3 Pt command 131
Arc Ctr command 131
arcs
  adding 129, 131
  circular interpolation 47, 174
  converting segments to 214
  converting to line segments 214
  copying 108
  D-codes 106, 115
  deleting 106, 110
  mirroring 114
  moving 106, 109

arcs *(con't)*
polarity 106, 116
properties 30, 106
rotating 113
scaling 115
tangential 129
Arcs 360 command 174
Arcs To Segmented Arcs command 214
ARL files 47, 235
array, adding 131
Array command 131
arrows 155
Assembly menu 218
assembly reverse engineering 74
Automatic command 210
Automatic Conversion command 209
automatic conversion rule files 235
aperture lists 82
creating 82, 236
regular expression syntax 235
autopan 38, 124, 230

B

background color 35
Backside command 125
balloon text 154, 162
Barco DPF command 86, 97
batch printing 101
bed of nails 69, 200
bill of materials
importing 15, 224
printing report 15, 152, 223
bird's-eye view 35
Bitmap command 101
Blind/Buried command 145
blind/buried vias 145
block instance 30
BMP files, exporting 101
BOM command 152
BOM Report command 223
Break Tab command 119, 120, 137
break tabs
active 6
adding 137
changing 119
defining 148
deleting 120
moving 121
Break Tabs command 148
Build Embedded Passive command 206
bullet apertures 140
butterfly apertures 140

C

CAM file
importing 77, 88
merging 78
CAM350 77
CAM350 command 88
canceling commands 3
CCW Inside command 135
CCW Outside command 136
Center To Center command 173
centroid file, importing 221
Centroid File command 221
chamfer 113
Change command 119
Change Drill Tools command 229
chord angle 214
Circle command 130
Custom Aperture Editor 231
circles
adding 130
drilled 135
in custom apertures 231
milled 135, 136
properties 30
circular interpolation 47, 130, 131, 174, 214
Clear command 106
Clear Highlights command 126
Clear Netlist command 188
click 40
Clip command 112
Clipboard 43, 106
Close button 3
Close command 78
color
layer 36
redlining 156
tool 146
Color Bar 36
color list file 243
Color Selector 36
customizing 36
color.rgb 243
Combine Paths command 122
Commands tab 27
comments 153, 154, 162
compare
layers 22
netlists 65, 89, 90, 91
compensation 117, 146, 196
etched passive 206
Composite To Layer command 217
composites 13, 47, 60
adding 13, 144
adding layers to 13
converting to layers 13, 217
deleting 13, 144
editing 13, 144
hiding in workspace 13
viewing 13
Composites command 125, 144
Configure command 174
construction line
adding 160
deleting 160
editing 160
Convert command 209
coordinate format 47
coordinates 35
changing for items 106
copper area 169
Copper Area command 169
copper slivers 164

Copy command 43, 108
  Custom Aperture Editor 228
Copy Selection To Clipboard command 43, 106
coupons 67, 182
Create Path From command 196
cross-hatch pattern 182
crown break tab 148
Current Folder 34
cursor, snapping to grid 174
Custom Aperture Editor 140
custom apertures 10, 47, 140
  adding circles to 231
  adding lines to 231
  assigning to NC tools 146
  converting draws/flashes to 215
  creating 10
  editing 10, 228, 229
  exploding 115
  in 274-D aperture lists 233
  saving 228
  scaling 10
  viewing outline of 230
Customize Toolbar command 178
CVG file
  exporting 98
  importing 89
CW Inside command 136
CW Outside command 136

D

Data tab 7
Database Modified Indicator 34
D-Code command 115
D-codes 47, 57, 140, 233
  active 6
  changing 106
  mirroring 114
  polarity 47, 116
  redlining 156
  rotating 113
  scaling 115
  selection filter 29
  transcoding 115
  use report 151
delete
  analysis runs 22
  apertures 57, 140
  arcs 106, 110
  blind/buried layer sets 145
  break tabs 120
  composites 13, 144
  construction lines 160
  devices 15
  dimensions 159
  draws 106, 110
  drill charts 161
  drill hits 106, 110
  drill slots 106, 110
  drill tools 146
  drilled circles 106, 110
  drilled text 106, 110
  fabrication note balloons 163
delete *(con't)*
  fiducials 15
  flashes 106, 110
  footprints 15, 218
  isolated pads 195
  layer sets 145
  layers 139
  mill tools 146
  milled circles 106, 110
  NC tools 146
  netlists 12, 188
  operator messages 120
  optional stops 120
  part instances 15, 223
  polygons 106, 110
  redline information 156
  segments 112
  text 106, 110
  user data 106
  vertices 112
Delete command 110, 112, 120, 128, 156, 223
  Custom Aperture Editor 229
Delete Dimension command 159
Delete Hole Chart command 161
Delete Line command 160
Delete Note Balloon command 163
demo.mac 36
depth offset 146
design
  creating 44
  opening/importing 44
  saving/exporting 44
Design for Fabrication 164
Design Rule Check 164
desktop 3
Developer command 181
devices
  adding 218
  deleting 15
  highlighting 15
DFF 64, 164
dialog boxes 41
diamond apertures 140
Dimensioning command 157
dimensions
  adding 157
  adding construction lines 160
  deleting 159
  deleting construction lines 160
  editing 159
  editing construction lines 160
  properties 159
Disable Prompts button 34
Display Settings command 200
donut apertures 47, 140
dot pattern 182
Double Mill Path command 200
double stitched break tab 148
double-click 40
DPF files
  exporting 97
  importing 79, 86

Draw command 129
Drawing command 157
Drawn Pads command 209
Drawn Text command 211
draws 47
  adding 129
  converting mill data to 215
  converting to custom apertures 215
  converting to flashes 210
  converting to mill paths 108, 196, 215
  converting to polylines 216
  converting to text 211
  copying 108
  D-codes 106, 115
  deleting 106, 110
  marking as resistors 204
  mirroring 114
  moving 106, 109
  polarity 106, 116
  properties 30, 106
  rotating 113
  scaling 115
  use report 151
Draws To Polylines command 216
DRC 22, 64, 164
DRC Status 34
DRC/MRC command 164
drill chart
  creating 160
  defining drill legends 11, 146
  deleting 161
  modifying 161
  updating 162
Drill command 134, 160
drill files
  exporting 96
  importing 79, 84
drill hits
  adding 134
  converting flashes to 108, 215
  converting to flashes 108, 215
  converting to mill paths 217
  copying 108
  deleting 106, 110
  highlighting path 200
  mirroring 114
  moving 106, 109, 197
  properties 106, 117
  rotating 113
  snapping pads to 116
  tools 106, 117, 119
drill mask 198
Drill Mask command 198
Drill Properties command 117
Drill Separate command 197
drill slots 134
  converting to draws 215
  converting to mill paths 217
  copying 108
  deleting 106, 110
  exploding 121
  mirroring 114
drill slots *(con't)*
  moving 106, 109
  properties 30, 106
  rotating 113
  tools 106
Drill To Mill command 217
drill tool list
  exporting 97
  importing 85
drill tools 11
  adding 11, 146
  changing 117, 119
  deleting 11, 146
  editing 11, 146
  in custom apertures 229
  setup for test fixtures 200
  use report 151
drilled circles
  converting to draws 215
  converting to milled circles 217
  copying 108
  deleting 106, 110
  exploding 121
  mirroring 114
  moving 106, 109
  properties 30, 106
  rotating 113
  tools 106
Drilled command 135
drilled text
  adding 136
  copying 108
  deleting 106, 110
  editing 111
  exploding 121
  mirroring 114
  moving 106, 109
  properties 30, 106
  rotating 113
  tools 106
Drilled Text command 136
duplicate data 22, 169
DXF command 87, 98
DXF files
  exporting 98
  importing 79, 87

E

Edge To Edge command 172
edit
  apertures 57, 140
  arcs 106
  blind/buried layer sets 145
  break tabs 119, 121
  composites 60, 144
  construction lines 160
  custom apertures 10
  dimensions 159
  draws 106
  drill charts 161
  drill hits 106
  drill slots 106

edit *(con't)*
drill tools....146
drilled circles....106
drilled text....106, 111
fabrication note balloons....163
flashes....106
layer sets....145
layers....139
mill paths....106
mill tools....146
milled circles....106
NC tools....146
operator messages....119
polygons....106
redo....104
text....106, 111
undo....104
user data....106
vertices....112
Edit menu....104
ellipse apertures....140
e-mail design files....103
Embedded Passive command....171
embedded passives....20
defining....20, 204, 206
errors....22, 164
highlighting....20
marking resistors....20, 205
querying....171
report....151
viewing....20
viewing properties....30
Embedded Passives command....151, 204
Enable Prompts button....34
environment statement....236
Error ID....22
errors....22
analysis....22
fixing....22
highlighting....22
Esc key....3, 43
Etch Compensation command....206
Exit command....103
Custom Aperture Editor....228
Explode command....121
Explode Customs command....115
export....44
aperture lists....57, 95
Barco DPF....97
bed of nails test files....203
bill of materials....15
bitmaps....101
composites....60
DXF....98
FATF files....15, 225
fixture plate drill files....203
flying probe test files....202
Gerber....94
GerbTool....99
HPGL....99
IPC-2581....98
IPC-D-350....97
export *(con't)*
IPC-D-356....97
IPC-D-356A....97, 202, 203
NC files....96
NC tool lists....97
netlists....188
ODB++....99
ODB++(X)....99
Offspring....98
PostScript....100
Export command....94
Export FATF command....225
Extents command....173
external net....65, 168, 188
highlighting....13

F

Fabrication command....162
fabrication note balloon
deleting....163
editing....163
FATF file, exporting....15, 225
feature list....1
feed rate....117, 146
fiducials....67
deleting....15
highlighting....15
identifying....15, 224
printing report....15, 152, 224
Fiducials command....152, 224
fillet....113
film....207
film box, viewing extents....124
Film Box command....124
Find command....204
Find Duplicates command....169
fine pitch pads....188, 189, 190, 191
Fix SilkScreen command....194
fixture plate....69, 203
drill files....200, 203
Flash command....129
flashes....47
adding....129
converting draws to....210
converting drill hits to....108, 215
converting to custom apertures....215
converting to drill hits....108, 215
copying....108
D-codes....106, 115
deleting....106, 110
marking as resistors....204, 205
mirroring....114
moving....106, 109
polarity....106, 116
properties....30, 106
rotating....113
scaling....115
use report....151
Flatten command....185
flood fill....132
flying probe....69, 200
Flying Probe command....202

font files 241
footprint library 15, 150, 218
Footprint Library command 150, 218, 219
footprints
  creating 15, 150, 218
  deleting 15, 218
  highlighting 15
format statement 47
function keys 40, 43

G

galvanic venting 182
G-codes 47
Generate command 187, 188, 190, 201
Gerber 47
  converting NC data to 108, 215
  converting to NC data 108, 215
  exporting 94
  importing 79, 81
Gerber command 81, 94
Gerber To NC command 215
GerbTool command 99
GerbTool-Stroke font 133, 241
green light 34
grid 35
  size 203, 231
  snap 174, 203, 231
  viewing 125
Grid command 125
  Custom Aperture Editor 231
Grid Snap command 174
group order 198
group selection 105
GTD file
  merging 78
  opening 77

H

hatch
  polygon fill 132
  vent pattern 182
Help button 41
hexagon apertures 140
highlight
  drill paths 200
  errors 22
  item types 172
  nets 170
  removing 126
  select groups 126
  turning on/off 126
  User Data 171
Highlight command 172
Highlights command 126
hole chart
  creating 160
  deleting 161
  drill legends 11, 146
  modifying 161
  updating 162
home plate 207
hotkeys 38, 43
HPGL
  exporting 99
  importing 79, 87
  printing 99
HPGL command 87, 99
HPGL/2 87

I

Identify command 224
Identify Using command 219
import 44
  aperture lists 57, 79, 82
  Barco DPF 79, 86
  bill of materials 15, 224
  CAM350 ....77, 88
  centroid file 221
  composites 60
  drill 79, 84
  DXF 79, 87
  Gerber 79, 81
  HPGL 79, 87
  HPGL/2 ....79, 87
  IPC-2581 89
  IPC-D-356(A) 92
  mill 79, 96
  NC 79, 84
  NC tool lists 85
  ODB++ 89
  ODB++ netlists 93
  ODB++(X) 90
  ODB++(X) netlists 93
  Offspring 89
  PADS ASCII 91
  PADS ASCII netlists 94
Import BOM command 224
Import command 79
Import Settings command 179
Import Wizard 79
Import Wizard command 79
incremental coordinates 47
intrinsic apertures 140
Invert command 106
IPC-2581
  exporting 98
  importing 89
IPC-2581 (Offspring) command 89, 98
IPC-D-350, exporting 97
IPC-D-350 command 97
IPC-D-356 command 92, 97
IPC-D-356 Data command 219
IPC-D-356(A) 69
  exporting 97
  exporting for test 202, 203
  importing 92
IPC-D-356A command 202
isolated
  pads 195
  thermals 164
Isolated command 195
Item command 106, 170
  Custom Aperture Editor 231
Item Properties 30

J

Join command 113

L

Layer Bar 6
Layer Compare command 168
layer sets 14
  adding 14, 145
  blind/buried vias 145
  deleting 14, 145
  editing 14, 145
  MCM/LTCC 145
  viewing 14, 145
Layer Sets command 145
Layer Spread command 207
layers 7, 36
  active 6, 7, 36
  adding 7, 139
  adding to composites 13
  adding to layer sets 145
  aligning 116
  blind/buried layer sets 145
  color 36
  comparing 168
  composites 47, 144
  converting composites to 217
  copper area 169
  deleting 7, 139
  editing 7, 139
  LTCC stackup 145
  MCM stackup 145
  measuring 173
  mirroring 114
  name 47, 139
  offsetting 114
  polarity 47
  registration 116, 164
  removing from composites 13
  rotating 113
  scaling 114
  selection filter 29
  type 36, 139
  viewing redlines 157
  visibility 7, 36
Layers command 139
library files 140
Line command 231
linear interpolation 47
lines
  adding redline 155
  adding to custom apertures 231
  adding vertices to 111, 130
  converting to arcs 214
  converting to polylines 216
  deleting segments 112
  deleting vertices 112
  joining 113
  orthogonal snap 174
link tool 146
Load command 181
log, saving 103
Log tab 127
LTCC stackup 145

M

M.N 47
Macro Developer 181
macros 27, 29
  loading 181
  recording 181
  running 29, 181
Main tab 127
main window 3
Manual Conversion command 208
Manufacturing Rule Check 164
Mark Resistor Flashes command 205
mask slivers 164
max hits 146
Maximize button 3
MCM stackup 145
MCM/LTCC Stackup command 145
M-codes 47
measure
  center-to-center 173
  copper area 169
  edge-to-edge 172
  extents 173
  point-to-point 172
Measure command 172
memory 43
  purging 122
Menu bar 3
merge
  CAM files 78
  GTD files 78
  PCB files 78
  VCAM files 78
Merge command 78
Metric command 232
mill files
  exporting 96
  importing 79, 84
Mill Path command 134
mill paths
  adding 134, 196
  converting draws to 108, 196, 215
  converting to draws 108
  converting to drills 217
  converting to polylines 215
  copying 108
  deleting 106, 110
  deleting segments 112
  double-routing 200
  exploding 121
  joining 113
  mirroring 114
  moving 106, 109
  plunge point 120
  properties 30, 106, 117
  rectangle 130
  reversing direction 121
  rotating 113
  tools 106, 117, 119
  vertices 111, 112, 130

Mill Plunge Point command....120
mill tabs....137, 148
Mill To Drill command....217
mill tool list
  exporting....97
  importing....85
mill tools....11
  adding....11, 146
  changing....117, 119
  deleting....11, 146
  editing....11, 146
  use report....151
milled circles
  adding....135, 136
  converting to drilled circle....217
  converting to polylines....215
  copying....108
  deleting....106, 110
  exploding....121
  mirroring....114
  moving....106, 109
  properties....30, 106
  rotating....113
  tools....106
Milled command....135
Minimize button....3
Mirror command....114
mirror image....47
Modify Dimension command....159
Modify Hole Chart command....161
Modify Line command....160
Modify Note Balloon command....163
moire apertures....47, 140
mouse functions....40, 43
Move Break Tab command....121
Move command....109, 112
  Custom Aperture Editor....228
MRC....164
MRU file list....103
My Commands....27

N

Navigator....7
  Analysis....22
  Apertures....8
  Assembly....15
  Commands tab....27
  Composites....13
  Custom Apertures....10
  Data tab....7
  Embedded Passives....20
  External Nets....13
  hiding....127
  Layer Sets....14
  Layers....7
  Macros....29
  My Commands....27
  NC Tools....11
  Nets....12
  System Commands....28
  viewing....127
Navigator command....127
NC (Drill/Mill) command....84, 96
NC Circle command....135
NC command....117, 195
NC files
  exporting....96
  importing....84
NC To Gerber command....215
NC tool list
  exporting....97
  importing....85
NC Tool List command....85, 97
NC Tool Table....146
NC tools....11
  active....6
  adding....11, 146
  changing....119
  deleting....11, 146
  editing....11, 146
  use report....151
NC Tools command....146, 151
negative polarity....47
nested commands....38, 43
Net command....170
net names....12, ,168, 188
  changing....12
Netlist command....91, 187
Netlist Compare command....65, 168
netlists....12, 69, 187
  clearing....12, 188
  comparing....65, 89, 90, 91, 164, 168
  exporting....97, 202, 203
  importing....92, 93, 94
  sample....242
  saving....188
nets....12
  adding user data to....12
  changing items in....106
  changing names....12
  defining for in-circuit test....188
  highlighting....12, 170
  querying....12, 170
  viewing items in....106
New command....44, 77
New Group command....105
NLC....164
notes....153, 154, 162

O

oblong apertures....140
octagon apertures....140
ODB++
  exporting....99
  importing....89
  intrinsic apertures....140
  netlist....93
ODB++ command....89, 93, 99
ODB++(X)
  exporting....99
  importing....90
  netlist....93
ODB++(X) command....90, 93, 99
offset, layers....114

Offspring
exporting....98
importing....89
open....44
aperture lists....82
Barco DPF....79, 86
CAM files....77
CAM350....77, 88
drill....79, 84
DXF....79, 87
Gerber....79, 81
GTD files....77
HPGL....79, 87
IPC-2581....89
IPC-D-356(A)....92
mill....79, 84
NC....79, 84
NC tool lists....85
ODB++....89
ODB++(X)....90
Offspring....89
PCB files....77
VCAM files....77
Open command....44, 77
opens....164, 168
operator message
adding....138
changing....119
deleting....120
Operator Message command....119, 120, 138
Optimize command....189, 191, 199
optional stop
adding....138
deleting....120
Optional Stop command....120, 138
origin, changing....117
Origin command....117
Ortho Line Snap command....174
orthogonal snap....174
outline view....125
Overlay command....125
overshoot....117, 196

P

Pad Removal command....195
pads
converting draws to flashes....210
isolated....195
redundant....195
snapping....116
stacked....195
PADS ASCII command....91, 94
Page Setup command....101
Pan command....124, 230
panelization....67, 182, 185
Panelize command....182
panels
flattening....185
image properties....30
origin....182
templates....67
parts
deleting....15, 223
highlighting....15
identifying....15, 219, 220, 221, 222
identifying fiducials....15
querying....15, 223
Parts command....218
Paste Clipboard command....106
paste mask
creating....190
errors....22
optimizing....191
Paste Mask command....190
Path Properties command....117
pilot tool....146
pin holes....164
pin numbers....168
pinning holes....182
plating....117, 198
plunge point....196
changing....117, 120
Point To Point command....172
polarity....47, 125, 144
changing....106, 116
converting mixed to positive....217
Polarity command....116
Polygon command....132
polygons....47, 132
adding....132
area fill....47
converting raster to vector....216
copying....108
D-codes....106, 115
deleting....106, 110
mirroring....114
moving....106, 109
polarity....106, 116
properties....30, 106
rotating....113
scaling....115
voids....170
polylines
converting draws to....216
converting mill data to....215
positive polarity....47
PostScript
exporting....100
printing....100
PostScript command....100
pour....132
PowerPCB....91
preferences....174
Previous command....126
Print command....102
Print Preview command....103
Print Setup command....103
printing....102
HPGL....99
layer spread....207
PostScript....100
preview....103
setup....101, 103

probes 203
Process Indicator 34
Properties command 156, 159
properties display 30
Purge command 122

Q

query
  custom aperture elements 231
  database items 170
  embedded passives 171
  nets 170
  parts 15, 223
  User Data 171
Query command 207, 223

R

raster fill 132
Raster To Vector command 216
Recall command 126
Record command 181
Rectangle command 130
rectangles
  adding 130
  aperture 140
red light 34
Redline command 153
redlining 153
  arrows 155
  colors 156
  D-codes 156
  deleting 156
  lines 155
  properties 156
  sketches 156
  text 153
  text in balloon 154
  viewing 157
Redo command 104
redraw, interrupting 35, 43
Redraw command 125, 230
Redraw Status 34
redundant data 169
redundant pad removal 195
reference designators 168
Rel button 35
relative coordinates 35
Remove From command 105
Report command 207, 224
reports
  analysis 22
  apertures 151
  bill of materials 15, 152, 223
  D-codes 151
  draws 151
  embedded passives 151
  fiducials 15, 152, 224
  flashes 151
  NC tools 151
Reports command 151
resist slivers 164
resistors, marking flashes as 204, 205
Restore Toolbars command 127
retract rate 146
reverse engineering 74
Reverse Path Direction command 121
right-click 40, 43
robber bar 67, 182
Rotate command 113
round apertures 140
RPM 146
rule statement 236
Run command 181

S

save 44
  aperture lists 95
  Barco DPF 97
  bed of nails test files 203
  bitmaps 101
  custom apertures 228
  DXF 98
  fixture plate drill files 203
  flying probe test files 202
  Gerber 94
  GerbTool 99
  HPGL 99
  IPC-2581 98
  IPC-D-350 97
  IPC-D-356(A) 97, 202, 203
  logs 103
  NC files 96
  NC tool lists 97
  netlists 188
  ODB++ 99
  ODB++(X) 99
  Offspring 98
  postscript 100
  VCAM files 79
  VisualCAM database files 79
Save As command 44, 79
Save command 44, 79, 126, 188
  Custom Aperture Editor 228
Save Files command 203
Save Log command 103
Scale command 114, 115
scaling
  custom apertures 10
  D-codes 115
  layers 114
Segment Delete command 112
Segmented Arcs To Arcs command 214
segments, deleting 112
Select command 105
Select Draws command 210
Select Footprint command 222
select group 34, 105
  adding to 105
  changing items in 106
  clearing 106
  creating 105
  highlighting 126
  inverting 106
  removing items from 105
  viewing items in 106

Selection Filter....29, 43
viewing....126
Selection Filter command....126
Selections command....126
Send command....103
SeqNo....30, 106
sequence number....106
Set Group Order command....198
Set Netlist Information command....188
Setup command....200
Setup Shapes command....207
shorts....164, 168
SHX....87
silkscreen....194
SilkScreen Layer command....220
Simple command....185
Sketch command....125, 156
Custom Aperture Editor....230
slivers....164
Slot command....134
slots
adding....134
deleting....106, 110
mirroring....114
moving....106
properties....30, 106
rotating....113
tools....106
SMD pitch....164
SMD spacing....164
SMT....140
Snap Pads command....116
Snap To Grid command....203
Snoman command....193
solder bridges....164
solder mask
creating....188
errors....22
optimizing....189
Solder Mask command....188
square apertures....140
Stacked command....195
stacked pad removal....195
Stagger command....202
starburst pattern....182
starved thermals....164
Status Bar command....231
Stencil Shapes command....150
stencils, creating....73
Stencils command....207
step & repeat....47, 67
removing from panels....185
Stick mode....125
stitched break tab....148
stitched crown break tab....148
stop
adding....138
deleting....120
System Commands....28
system requirements....1

T

tabs
adding....137
changing....119
defining....148
deleting....120
moving....121
target apertures....140
teardrops....194
Teardrops command....194
tenting....198
terminating commands....3
Test Fixture command....203
test fixturing....69, 200
creating test points....201
setup tools....200
staggering test points....202
test points....69, 168, 201
adding/deleting individually....201
analyzing....202
creating....200
staggering....202
Test Points command....200
test probes....69, 203
text
adding....133
converting draws to....211
copying....108
D-codes....106, 115
deleting....106, 110
drilled....136
editing....111
font files....241
mirroring....114
moving....106, 109
polarity....106, 116
properties....30, 106
redline....153, 154
rotating....113
scaling....115
Text command....111, 133
thermal apertures....47, 140
thieving....67, 182
thru hole apertures....140
title blocks....67, 182
To Custom command....215
tool lists....146, 200
exporting....97
importing....85
Toolbar command....230
toolbars
customizing....127, 178
importing settings....179
restoring display....127
Toolbars command....127
tooling origin....182
tooling path....198
optimizing....199
tools....11
changing....106, 117, 119
number....146
selection filter....29

tools *(con't)*
table 146
use report 151
Tools command 119
Transcode command 115
transparency 125
triangle apertures 140
TrueType font 133
tutorials
comparing netlists 65
copying to the Windows Clipboard 43
creating stencils 73
Gerber for beginners 47
panelization 67
performing DRC & DFF analyses 64
reverse engineering for assembly 74
test fixturing 69
using files 44
working with apertures 57
working with composites 60
working with embedded passives 72

U

Undo command 43, 104
Custom Aperture Editor 228
purging buffer 122
reversing 104
Undo Status 34
units of measure 174
Custom Aperture Editor 232
Units/Precision command 174
Update Hole Chart command 162
user data
adding 106
adding to nets 12
assigning net/pin data to 168
changing 106
deleting 106
querying 171
viewing 106
User Data command 171
User menu 226

V

VCAM file
merging 78
opening 77
saving 79
vector fill 132
venting 67, 182, 185
Vertex command 111, 130
vertices
adding 111, 130
deleting 112
vertices *(con't)*
moving 112
View Bar command 231
View menu 123
View Redlining command 157
View Tabs 35
adding 127
deleting 128
View Tabs command 127
View/Edit command 145
viewing
autopan 124, 230
backside 125
composites 60, 125
database extents 124, 173
film box extents 124
grid 125
highlights 126
outlines 125
overlay 125
panning 124, 230
polarity 125
recalling previous 126
recalling saved 126
redlines 157
refreshing 125, 230
saving current 126, 127
select group 126
sketch mode 125
stick mode 125
transparent 125
user data 106
virtual panels 126
zooming 123, 124, 229, 230
virtual layer 185
virtual memory 43
Virtual Panel command 126
virtual panelization 185
VisualCAM net 65, 168

W

Window command 123, 229
window selection 29
Workspace
clearing 77
refreshing display 125, 230

X

XY Bar 35

Z

zero suppression 47
zoom 123, 124, 229
Zoom In command 123, 229
Zoom Out command 124, 230